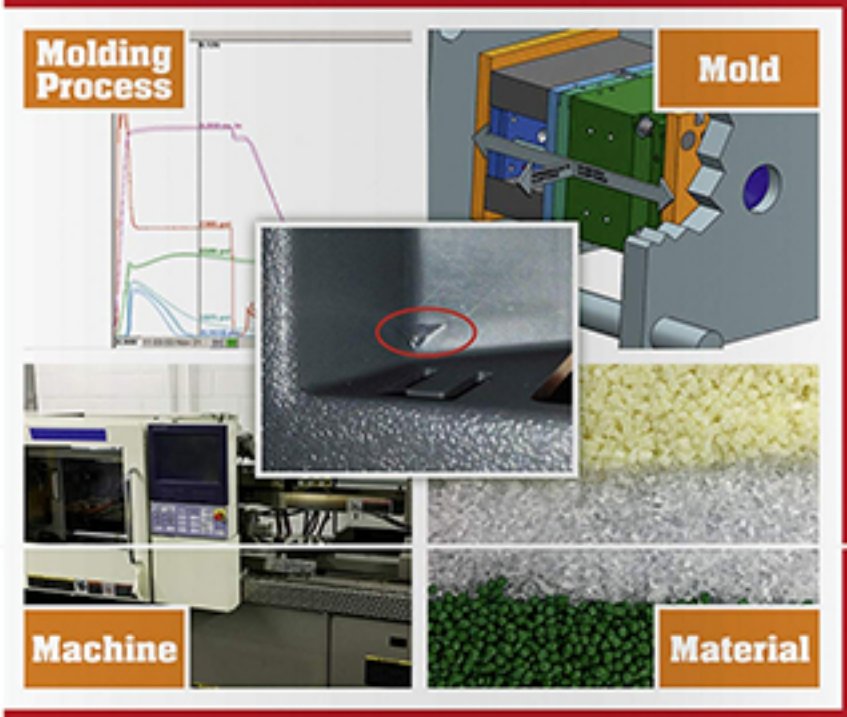
Randy Kerkstra
Steve Brammer

Injection Molding

Advanced Troubleshooting Guide

The 4M Approach



eBook Bonus

HANSER

Kerkstra / Brammer

Injection Molding Advanced Troubleshooting Guide

Randy Kerkstra
Steve Brammer

Injection Molding Advanced Trouble-shooting Guide

HANSER

Hanser Publishers, Munich

Hanser Publications, Cincinnati

The Authors:

Randy Kerkstra, Hudsonville, MI; info@kbmoldingsolutions.com

Steve Brammer, Holland, MI; info@kbmoldingsolutions.com

Distributed in the Americas by:
Hanser Publications
6915 Valley Avenue, Cincinnati, Ohio 45244-3029, USA
Fax: (513) 527-8801
Phone: (513) 527-8977
www.hanserpublications.com

Distributed in all other countries by:
Carl Hanser Verlag
Postfach 86 04 20, 81631 München, Germany
Fax: +49 (89) 98 48 09
www.hanser-fachbuch.de

The use of general descriptive names, trademarks, etc., in this publication, even if the former are not especially identified, is not to be taken as a sign that such names, as understood by the Trade Marks and Merchandise Marks Act, may accordingly be used freely by anyone. While the advice and information in this book are believed to be true and accurate at the date of going to press, neither the authors nor the editors nor the publisher can accept any legal responsibility for any errors or omissions that may be made. The publisher makes no warranty, express or implied, with respect to the material contained herein.

The final determination of the suitability of any information for the use contemplated for a given application remains the sole responsibility of the user.

Cataloging-in-Publication Data is on file with the Library of Congress

All rights reserved. No part of this book may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying or by any information storage and retrieval system, without permission in writing from the publisher.

© Carl Hanser Verlag, Munich 2018
Editor: Dr. Mark Smith
Production Management: Jörg Strohbach
Coverconcept: Marc Müller-Bremer, www.rebranding.de, München
Coverdesign: Stephan Rönigk
Typesetting: Kösel Media GmbH, Krugzell
Printed and bound by Printed and bound by Druckerei Hubert & Co GmbH und Co KG BuchPartner, Göttingen
Printed in Germany

ISBN: 978-1-56990-645-3
E-Book ISBN: 978-1-56990-646-0

Preface

Why a book dedicated to troubleshooting? The answer to that lies in frustrations that the authors have experienced over the years in launching and maintaining injection molds and processes for those molds. Have you ever experienced any of the following?

- High scrap rates
- Excess down time
- Slow cycle times
- Customer rejections (both internal and external)
- Processing around tooling issues
- Damage to molds
- Defects that seem to show up out of nowhere
- Defects that keep reoccurring
- "Fixed" problems that keep coming back
- Molds that run fine in one machine but not in another

The goal of this book is to help provide tools and information that will help truly address these types of concerns.

Often times in the plastics industry there is a great deal of learning through the "school of hard knocks" and both of the authors have had the opportunity to learn a lot this way. What the industry has not done well is passing along these lessons to others. We have reached a point where we are interested in passing along the lessons we have learned through experience in the trenches.

One thing that the authors have noticed in the industry is a disconnect between processing and tooling. We come at troubleshooting with different backgrounds, one in tooling and one in materials and processing. However, we both approach a problem with the same thought process. We felt it was time to provide a resource that dives deep into the interaction of tooling, processing, and materials and have sought to create that in this work. There would not be a value in this book if it was just regurgitation of the same information that has been documented in

"troubleshooting guides" over the years. The strength of this book lies in combining hands-on experience of making molds and processes run successfully even and especially when the "by-the-books" approach did not resolve the problem.

This book is broken down into specific sections:

1. Troubleshooting methodology and tools (Chapters 1 and 2)
2. Focused discussion of key areas impacting troubleshooting including the mold, machine, material, and molding process (Chapters 3–14)
3. In-depth alphabetical troubleshooting guide for various defects (Chapters 15–47)

   Again the key differences in this troubleshooting guide come down to the efforts to bridge the gaps between tooling, processing, and materials and provide in-depth feedback from designing, building, processing, maintaining, and troubleshooting 1000s of molds over the last 25+ years.

Writing this book was a humbling experience. We obviously have not seen every mold, material, and machine combination and could come up with "what if?" cases forever, but we feel this book well represents our experience. We believe that keeping an open mind to solutions is critical to successful troubleshooting. We would never believe that we have all the answers. Remember there is always more to learn!

Many of the fundamental processing and design methods that are the industry standards have come from the development and training efforts of many individuals including Rod Groleau, John Bozzelli, Don Paulson, Glenn Beall, and John Beaumont. Steve was fortunate to have Richard Brammer, a mold maker, development engineer, and Ferris State University Instructor as a father who provided guiding influence along the "plastics road". Steve is also thankful for the Plastics Engineering Instructors at Ferris State University and Grand Rapids Community College. The above people have been educating the plastics industry for many years and through their writing and instruction the industry has learned and improved. We have also had the opportunity to work with many excellent processors, maintenance people, mold builders, designers, and material scientists who have added to our learning over our careers. We would also like to thank Mark Smith and the team at Hanser Publishers for helping bring this book to reality.

As you move down the troubleshooting road keep learning, always ask why, never assume, and stay open minded!

Randy Kerkstra

Steve Brammer

About the Authors

Randy Kerkstra has worked in the plastics industry for over 29 years, with specialism in troubleshooting injection molding. He also has much experience with injection molds, with 14 years in the tool shop environment as a mold maker/designer and over 15 maintaining and repairing/troubleshooting thousands of molds in the production environment. He has years of research with gate geometry, runner/sprue waste, and reducing part defects with a focus on the mold and how it impacts these issues. He also co-owns KB Molding Solutions, a training and consulting company. He currently works in sales and product development for PCS Company.

Steve Brammer has held a variety of positions for multiple custom and captive molders working in the furniture, appliance, automotive, and consumer products markets. He is currently Molding Technical Manager for a Tier One automotive component supplier. Steve is also an Instructor at Grand Rapids Community College, teaching courses in applied injection molding, plastics processing, and manufacturing. He also co-owns KB Molding Solutions, a training and consulting company. Steve has a Bachelor of Science degree in Plastics Engineering Technology from Ferris State University.

Contents

Preface ..... **V**

About the Authors ..... **VII**

1 Troubleshooting Methodology **1**

1.1 Troubleshooting 1

1.2 What Makes an Effective Troubleshooter? 1

1.3 What Makes an Ineffective Troubleshooter? 3

1.4 Troubleshooting Methodology 3

1.4.1 STOP: Systematically 4

1.4.2 STOP: Think 6

1.4.3 STOP: Observe 7

1.4.4 STOP: Proceed 10

1.4.5 STOP: Troubleshooting Cycle 12

1.4.6 Hard Fix versus Processing around Problem 13

1.4.7 Troubleshooting Tools 13

1.4.8 Troubleshooting Methodology Summary 18

2 Troubleshooting Tool Kit **21**

2.1 Lockout/Tagout 21

2.2 Hand Tools 21

2.3 Pyrometer 22

2.4 Spotting Blue 22

2.5 Measurement Tools 22

2.6 Multimeter 22

2.7 Process Monitoring Equipment 23

2.8 Moisture Analyzer 23

2.9 Dew Point Meter 23
2.10 Flashlight 24
2.11 Microscope/Magnifying Glass 24
2.12 Silly Putty 24
2.13 Inspection Mirror 24
2.14 Thermal Imaging Camera 25
2.15 Aluminum Tape 25
2.16 Dial Indicator 25
2.17 Purging Compound 25
2.18 Grinder/Stones 26
2.19 Camera 26
2.20 Material Data 26
2.21 Scale 26
2.22 Flow Meter 27
2.23 Mold Cleaning Supplies 27
2.24 Miscellaneous Supplies 27

3 Decoupled® or Scientific Molding 29

4 Gating Details 33
4.1 Gating 33
4.2 Gate Size, Shape, and Taper 35
4.3 Mental Picture Volume versus Pressure 40

5 Mold Hydraulics 41
5.1 Hydraulic Uses 41
5.2 Set and Pull Pressure 41
5.3 Cylinder Sizing 42
5.4 Hydraulic Cores or Slides 44
5.5 Multiple Sequences 44
5.6 Mounting 45
5.7 Switches 45

6 Mold Texture and Polish 47
6.1 Mold Texture 47
6.2 Mold Polish 50

7 Venting 53
7.1 Venting Basics 53
7.2 Alternate Venting 58
7.3 Venting Conclusion 60

8 Machine Performance 61
8.1 Load Sensitivity 63
8.2 Dynamic Check Valve Study 64
8.3 Velocity Linearity 65
8.4 10-Cycle Overlay 68
8.5 Cycle Variation 70
8.6 Velocity to Pressure Transfer 72
8.6.1 Machine Control 75
8.7 Machine Documentation 78

9 Drying 81
9.1 Introduction 81
9.2 Keys to Drying 81
9.2.1 Temperature 83
9.2.2 Dry Air 83
9.2.3 Air Flow 85
9.2.4 Time 85
9.3 Moisture Analysis 86
9.4 Material Handling 88
9.5 Material Concerns 90

10 Purging 91
10.1 Purging Impact 91
10.2 Purging Method Example 92
10.3 Purging Compounds 93

11 Hot Runners 95
11.1 Hot Runner Advantages 95
11.2 Heaters and Thermocouples 95
11.3 Hang Up Areas 96
11.4 Hot Drop Tips 96
11.5 Drooling, Stringing, and Sprue Sticking 97
11.6 Freeze Off 97
11.7 Orifice Size 97
11.8 Leakage 98
11.9 Zones and Wiring 99
11.10 Hot Runner Troubleshooting 99

12 Cavity Balance 101
12.1 Flow Length 103
12.2 Flow Diameter 103
12.3 Shear 103
12.4 Cooling 105
12.5 Venting 106
12.6 Clamp Pressure 106
12.7 Cavity Fill Balance Study 107
12.8 Artificial Balance 108
12.8.1 Family Molds 109

13 Cavity Instrumentation 111
13.1 Cavity Pressure Technology 111
13.2 Cavity Thermocouples 118
13.3 Process Documentation 121
13.4 Process Control 122
13.5 Additional Monitoring Options 123

14 Mold Cooling 125
14.1 Mold Cooling Importance 125
14.2 Water Flow 126
14.3 Documentation 127

14.4 Tooling Techniques 128
14.5 Other Cooling Concerns 130

15 Black or Brown Specks 131
15.1 Description 131
15.2 Black Specks Troubleshooting Chart 132
15.3 Black Specks Troubleshooting 132
15.3.1 Black Specks Troubleshooting Molding Process Issues 132
15.3.2 Black Specks Troubleshooting Mold Issues 133
15.3.3 Black Specks Troubleshooting Machine Issues 135
15.3.4 Black Specks Troubleshooting Material Issues 137

16 Blush 141
16.1 Description 141
16.2 Blush Troubleshooting Chart 142
16.3 Blush Troubleshooting 142
16.3.1 Blush Troubleshooting Molding Process Issues 142
16.3.2 Blush Troubleshooting Tooling Issues 145
16.3.3 Blush Troubleshooting Machine Issues 148
16.3.4 Blush Troubleshooting Material Issues 149

17 Brown Streaks 151
17.1 Description 151
17.2 Brown Streak Troubleshooting Chart 152
17.3 Brown Streak Troubleshooting 152
17.3.1 Brown Streak Troubleshooting Molding Process Issues 152
17.3.2 Brown Streak Troubleshooting Mold Issues 156
17.3.3 Brown Streak Troubleshooting Machine Issues 159
17.3.4 Brown Streak Troubleshooting Material Issues 162

18 Bubbles 165
18.1 Description 165
18.2 Bubbles Troubleshooting Chart 166
18.3 Bubbles Troubleshooting 166
18.3.1 Bubbles Troubleshooting Molding Process Issues 167
18.3.2 Bubbles Troubleshooting Mold Issues 168
18.3.3 Bubbles Troubleshooting Machine Issues 170
18.3.4 Bubbles Troubleshooting Material Issues 171

19 Buildup ... 173
19.1 Description ... 173
19.2 Buildup Troubleshooting Chart ... 174
19.3 Buildup Troubleshooting ... 174
19.3.1 Buildup Troubleshooting Molding Process Issues ... 174
19.3.2 Buildup Troubleshooting Mold Issues ... 175
19.3.3 Buildup Troubleshooting Machine Issues ... 177
19.3.4 Buildup Troubleshooting Material Issues ... 177

20 Burns ... 181
20.1 Description ... 181
20.2 Burns Troubleshooting Chart ... 182
20.3 Burns Troubleshooting ... 182
20.3.1 Burns Troubleshooting Molding Process Issues ... 182
20.3.2 Burns Troubleshooting Mold Issues ... 184
20.3.3 Burns Troubleshooting Machine Issues ... 186
20.3.4 Burns Troubleshooting Material Issues ... 189

21 Cloudiness ... 191
21.1 Description ... 191
21.2 Cloudiness Troubleshooting Chart ... 192
21.3 Cloudiness Troubleshooting ... 192
21.3.1 Cloudiness Troubleshooting Molding Process Issues ... 192
21.3.2 Cloudiness Troubleshooting Mold Issues ... 194
21.3.3 Cloudiness Troubleshooting Machine Issues ... 195
21.3.4 Cloudiness Troubleshooting Material Issues ... 196

22 Color Swirls ... 199
22.1 Description ... 199
22.2 Color Swirls Troubleshooting Chart ... 200
22.3 Color Swirls Troubleshooting ... 200
22.3.1 Color Swirls Troubleshooting Molding Process Issues ... 200
22.3.2 Color Swirls Troubleshooting Mold Issues ... 202
22.3.3 Color Swirls Troubleshooting Machine Issues ... 202
22.3.4 Color Swirls Troubleshooting Material Issues ... 204

23 Contamination ... 209
23.1 Description ... 209

23.2 Contamination Troubleshooting Chart 210
23.3 Contamination Troubleshooting 210
23.3.1 Contamination Troubleshooting Molding Process Issues 210
23.3.2 Contamination Troubleshooting Mold Issues 212
23.3.3 Contamination Troubleshooting Machine Issues 215
23.3.4 Contamination Troubleshooting Material Issues 217

24 Cracking 223
24.1 Description 223
24.2 Cracking Troubleshooting Chart 224
24.3 Cracking Troubleshooting 224
24.3.1 Cracking Troubleshooting Molding Process Issues 224
24.3.2 Cracking Troubleshooting Mold Issues 228
24.3.3 Cracking Troubleshooting Machine Issues 229
24.3.4 Cracking Troubleshooting Material Issues 230

25 Delamination 233
25.1 Description 233
25.2 Delamination Troubleshooting Chart 234
25.3 Delamination Troubleshooting 234
25.3.1 Delamination Troubleshooting Molding Process Issues 234
25.3.2 Delamination Troubleshooting Mold Issues 235
25.3.3 Delamination Troubleshooting Machine Issues 236
25.3.4 Delamination Troubleshooting Material Issues 237

26 Dimensions 239
26.1 Description 239
26.2 Dimensions Troubleshooting Chart 239
26.3 Dimensions Troubleshooting 240
26.3.1 Dimensions Troubleshooting Molding Process Issues 240
26.3.2 Dimensions Troubleshooting Mold Issues 245
26.3.3 Dimensions Troubleshooting Machine Issues 247
26.3.4 Dimensions Troubleshooting Material Issues 251

27 Excessive Cycle Time 255
27.1 Description 255
27.2 Excessive Cycle Time Troubleshooting Chart 255
27.3 Excessive Cycle Time Troubleshooting 256

27.3.1 Excessive Cycle Time Troubleshooting Molding Process Issues 256
27.3.2 Excessive Cycle Time Troubleshooting Mold Issues 259
27.3.3 Excessive Cycle Time Troubleshooting Machine Issues 262
27.3.4 Excessive Cycle Time Troubleshooting Material Issues 263

28 High Fill Pressure 265

28.1 Description 265
28.2 High Fill Pressure Troubleshooting Chart 265
28.3 High Fill Pressure Troubleshooting 266
28.3.1 High Fill Pressure Troubleshooting Molding Process Issues . . 266
28.3.2 High Fill Pressure Troubleshooting Mold Issues 267
28.3.3 High Fill Pressure Troubleshooting Machine Issues 269
28.3.4 High Fill Pressure Troubleshooting Material Issues 274

29 Flaking 277

29.1 Description 277
29.2 Flaking Troubleshooting Chart 278
29.3 Flaking Troubleshooting 278
29.3.1 Flaking Troubleshooting Molding Process Issues 278
29.3.2 Flaking Troubleshooting Mold Issues 278
29.3.3 Flaking Troubleshooting Machine Issues 280
29.3.4 Flaking Troubleshooting Material Issues 280

30 Flash 281

30.1 Description 281
30.2 Flash Troubleshooting Chart 282
30.3 Flash Troubleshooting 282
30.3.1 Flash Troubleshooting Molding Process Issues 283
30.3.2 Flash Troubleshooting Mold Issues 287
30.3.3 Flash Troubleshooting Machine Issues 292
30.3.4 Flash Troubleshooting Material Issues 296

31 Flow Lines 299

31.1 Description 299
31.2 Flow Lines Troubleshooting Chart 299
31.3 Flow Lines Troubleshooting 300
31.3.1 Flow Lines Troubleshooting Molding Process Issues 300

31.3.2 Flow Lines Troubleshooting Mold Issues 302
31.3.3 Flow Lines Troubleshooting Machine Issues 304
31.3.4 Flow Lines Troubleshooting Material Issues 306

32 Glass Fibers on Surface 309

32.1 Description 309
32.2 Glass Fibers on Surface Troubleshooting Chart 310
32.3 Glass Fibers on Surface Troubleshooting 310
32.3.1 Glass Fibers on Surface Troubleshooting Molding Process Issues 310
32.3.2 Glass Fibers on Surface Troubleshooting Mold Issues 312
32.3.3 Glass Fibers on Surface Troubleshooting Machine Issues 314
32.3.4 Glass Fibers on Surface Troubleshooting Material Issues 315

33 Gloss Variation 319

33.1 Description 319
33.2 Gloss Variation Troubleshooting Chart 320
33.3 Gloss Variation Troubleshooting 320
33.3.1 Gloss Variation Troubleshooting Molding Process Issues 320
33.3.2 Gloss Variation Troubleshooting Mold Issues 324
33.3.3 Gloss Variation Troubleshooting Machine Issues 327
33.3.4 Gloss Variation Troubleshooting Material Issues 327

34 Jetting 331

34.1 Description 331
34.2 Jetting Troubleshooting Chart 332
34.3 Jetting Troubleshooting 332
34.3.1 Jetting Troubleshooting Molding Process Issues 333
34.3.2 Jetting Troubleshooting Mold Issues 334
34.3.3 Jetting Troubleshooting Machine Issues 336
34.3.4 Jetting Troubleshooting Material Issues 336

35 Part Sticking on Cover 337

35.1 Description 337
35.2 Part Sticking on Cover Troubleshooting Chart 337
35.3 Part Sticking on Cover Troubleshooting 338
35.3.1 Part Sticking on Cover Troubleshooting Molding Process Issues 338

35.3.2 Part Sticking on Cover Troubleshooting Mold Issues 339
35.3.3 Part Sticking on Cover Troubleshooting Machine Issues 342
35.3.4 Part Sticking on Cover Troubleshooting Material Issues 343

36 Part Sticking on Ejector 345

36.1 Description 345

36.2 Part Sticking on Ejector Troubleshooting Chart 346

36.3 Part Sticking on Ejector Troubleshooting 346
36.3.1 Part Sticking on Ejector Molding Process Issues 346
36.3.2 Part Sticking on Ejector Troubleshooting Mold Issues 349
36.3.3 Part Sticking on Ejector Troubleshooting Machine Issues 351
36.3.4 Part Sticking on Ejector Troubleshooting Material Issues 352

37 Pin Push 355

37.1 Description 355

37.2 Pin Push Troubleshooting Chart 356

37.3 Pin Push Troubleshooting 356
37.3.1 Pin Push Troubleshooting Mold Process Issues 356
37.3.2 Pin Push Troubleshooting Mold Issues 358
37.3.3 Pin Push Troubleshooting Machine Issues 360
37.3.4 Pin Push Troubleshooting Material Issues 361

38 Read-through 363

38.1 Description 363

38.2 Read-through Troubleshooting Chart 364

38.3 Read-through Troubleshooting 364
38.3.1 Read-through Troubleshooting Molding Process Issues 364
38.3.2 Read-through Troubleshooting Mold Issues 365
38.3.3 Read-through Troubleshooting Machine Issues 367
38.3.4 Read-through Troubleshooting Material Issues 367

39 Scuffs and Scratches 369

39.1 Description 369

39.2 Scuffs and Scratches Troubleshooting Chart 370

39.3 Scuffs and Scratches Troubleshooting 370
39.3.1 Scuffs and Scratches Troubleshooting Molding Process Issues 370
39.3.2 Scuffs and Scratches Troubleshooting Mold Issues 373

39.3.3 Scuffs and Scratches Troubleshooting Machine Issues 375
39.3.4 Scuffs and Scratches Troubleshooting Material Issues 375

40 Short Shot 377

40.1 Description 377

40.2 Short Shot Troubleshooting Chart 378

40.3 Short Shot Troubleshooting 378
40.3.1 Short Shot Troubleshooting Molding Process Issues 378
40.3.2 Short Shot Troubleshooting Mold Issues 385
40.3.3 Short Shot Troubleshooting Machine Issues 389
40.3.4 Short Shot Troubleshooting Material Issues 393

41 Sink 397

41.1 Description 397

41.2 Sink Troubleshooting Chart 398

41.3 Sink Troubleshooting 398
41.3.1 Sink Troubleshooting Molding Process Issues 398
41.3.2 Sink Troubleshooting Mold Issues 404
41.3.3 Sink Troubleshooting Machine Issues 408
41.3.4 Sink Troubleshooting Material Issues 411

42 Splay 413

42.1 Description 413

42.2 Splay Troubleshooting Chart 414

42.3 Splay Troubleshooting 414
42.3.1 Splay Troubleshooting Molding Process Related Issues 415
42.3.2 Splay Troubleshooting Mold Issues 420
42.3.3 Machine Issues 425
42.3.4 Material Issues 427

43 Sprue Sticking 431

43.1 Description 431

43.2 Sprue Sticking Troubleshooting Chart 432

43.3 Sprue Sticking Troubleshooting 432
43.3.1 Sprue Sticking Troubleshooting Molding Process Issues 432
43.3.2 Sprue Sticking Troubleshooting Mold Issues 435
43.3.3 Sprue Sticking Troubleshooting Machine Issues 439
43.3.4 Sprue Sticking Troubleshooting Material Issues 441

44 Stringers 443

44.1 Description 443

44.2 Stringers Troubleshooting Chart 444

44.3 Stringers Troubleshooting 444

44.3.1 Stringers Troubleshooting Molding Process Issues 444

44.3.2 Stringers Troubleshooting Mold Issues 446

44.3.3 Stringers Troubleshooting Machine Issues 448

44.3.4 Stringers Troubleshooting Material Issues 450

45 Voids 451

45.1 Description 451

45.2 Voids Troubleshooting Chart 452

45.3 Voids Troubleshooting 452

45.3.1 Voids Troubleshooting Molding Process Issues 452

45.3.2 Voids Troubleshooting Mold Issues 454

45.3.3 Voids Troubleshooting Machine Issues 456

45.3.4 Voids Troubleshooting Material Issues 456

46 Warp 457

46.1 Description 457

46.2 Warp Troubleshooting Chart 457

46.3 Warp Troubleshooting 458

46.3.1 Warp Troubleshooting Molding Process Issues 459

46.3.2 Warp Troubleshooting Mold Issues 465

46.3.3 Warp Troubleshooting Machine Issues 467

46.3.4 Warp Troubleshooting Material Issues 468

47 Weld Lines 471

47.1 Description 471

47.2 Weld Lines Troubleshooting Chart 472

47.3 Weld Lines Troubleshooting 472

47.3.1 Weld Lines Troubleshooting Molding Process Issues 473

47.3.2 Weld Lines Troubleshooting Mold Issues 475

47.3.3 Weld Lines Troubleshooting Machine Issues 477

47.3.4 Weld Lines Troubleshooting Material Issues 478

Index 481

1 Troubleshooting Methodology

1.1 Troubleshooting

Troubleshooting is problem solving. Molding troubleshooters are called upon to resolve problems with the part, mold, machine, or process. There are many problems encountered in injection molding including these general categories:

- Cosmetic defects
- Dimensional problems
- Part breakage
- Long cycle times
- High scrap rate

All of the above lead to increased cost to manufacture a molded part, which often makes the difference between profit and loss. A molding operation that is consistently running high scrap or long cycles is going to struggle to succeed.

1.2 What Makes an Effective Troubleshooter?

The role of a troubleshooter is to find the root cause of a problem and do what is necessary to resolve the problem. Effective troubleshooters will look beyond their initial impressions and ensure that the true root cause has been addressed. Good troubleshooters take a great deal of pride in having the perseverance to solve a problem and ensure that it does not reoccur.

The Merriam-Webster dictionary defines a troubleshooter as:

A skilled worker employed to locate trouble and make repairs in machinery and technical equipment.

A person skilled at solving or anticipating problems or difficulties.

Troubleshooting is a skill that can be learned and this book is intended to help convey some of the knowledge that the authors have learned through many years of troubleshooting. Some of the key things that will help anyone improve at troubleshooting include:

- Willingness to listen to others. Anyone can provide the crucial piece of information that helps solve a problem. A good troubleshooter will listen to people.
- Being observant. A good troubleshooter will always be looking for what might have changed. Good observation skills are critical to troubleshooting. Good troubleshooters live by the motto "show me" rather than trusting that things have been set up correctly. Anyone who has spent time troubleshooting will tell you that there are plenty of cases where they were told that the material was dry or the mold was clean but verification showed otherwise.
- Willingness to learn. Many times when working on a problem a troubleshooter will have to dig deep into a subject to learn what the root cause really is. Be open to learning and use all resources available to become better at troubleshooting. There is always more to learn.
- Perseverance. This is critical to being a good troubleshooter. There are many times when standing at a molding machine for hours gets very tiring. A good troubleshooter is willing to put the time and effort in to ensure the problem is corrected. This also means that they will check back on the problem to ensure that it is corrected.
- Willingness to try things. If a troubleshooter is afraid to try something out of fear of a negative result they will struggle to reach the solution of the problem. A perfect example is a processor who is afraid to open up vents on a mold because of flash. If you do not try to fix the problem it will not be resolved.
- Taking a systematic approach. A good troubleshooter works through a problem using a systematic methodology. Change one thing at a time in an organized fashion and give the change a chance to stabilize.
- Being data driven. Good troubleshooters utilize data to make decisions, and do not rely on assumptions or opinions. If a change is made the data should provide feedback on the whether or not there was an improvement.
- Patience. This may be one of the hardest parts of troubleshooting. Often times a change is made but the troubleshooter is not patient enough to determine the effect and immediately makes another change. Allow processes to stabilize during troubleshooting to determine the ultimate impact.

1.3 What Makes an Ineffective Troubleshooter?

Many of the above characteristics help people to become effective troubleshooters. There are also many traits that make people struggle when troubleshooting including:

- The "know it all". People that believe they know everything about every aspect of injection molding will one day be in for a rude awakening. Injection molding problems tend to have a humbling effect on troubleshooters, and everyone has something more to learn. Remember every mold, machine, and material combination can create a new opportunity.
- The "this worked last time" syndrome. Many times people get caught in an approach that completely relies on what they have experienced, which in turn puts blinders on them. First understand the problem before trying to implement what worked last time.
- The "Band-Aids and duct tape fixes everything" troubleshooter. This type of person will always look for the simplest thing that can be done whether or not they solve the problem. This mentality often happens in production where the approach can be just "get me the parts I need to make shipment." While a "duct tape" type of fix may help to limp through a run, the root cause must be addressed and corrected. Putting "Band-Aids" on top of duct tape to keep a job running will lead to scrap and downtime.
- The "flavor of the month". This often happens when a specific problem is identified and corrected on a given mold in the plant. Often since this solution solved that problem people will try to implement that solution everywhere whether it fits or not.

Overall many people that struggle to effectively troubleshoot are lacking either the time or the tools to be successful. There is always only going to be 24 hours in every day and customer demand for quality parts will persist. This book was written to help provide some tools that can make troubleshooting more efficient and hopefully help people wisely use their time spent troubleshooting.

1.4 Troubleshooting Methodology

As mentioned in Section 1.2, a good troubleshooter uses a systematic approach. The following is a reminder to help with keeping a systematic approach to troubleshooting;

Systematically

Think

Observe

Proceed

This STOP methodology of troubleshooting is meant to do exactly what it says and stop before jumping to conclusions.

Development of STOP

This thought process came years ago while interviewing process engineers and technicians. I would always try to gauge their knowledge by asking questions about how they would handle a problem such as a short shot. The answers I received were usually correct to a point but obviously quite diverse. Often times the answers provided could be the right ones, but, without knowing what was happening, could also lead to disaster. When I reviewed my own mentality, I came to understand that the first thing I would do when troubleshooting was to stop and really examine what was happening. The concept of STOP troubleshooting came about as an easy way to train people in the methodology of troubleshooting.

1.4.1 STOP: Systematically

In the STOP methodology, the S stands for systematically. All troubleshooting should be conducted in an organized and systematic approach. Having a systematic approach will help ensure the root cause of the problem is truly resolved. As a problem is addressed a systematic approach will make it easier to avoid missing a potential cause.

Part of the systematic approach to troubleshooting breaks the problem into four key categories. Many people are familiar with the 5M's often used for fishbone diagrams which are man, method, machine, measurement, and material. For systematic injection molding troubleshooting the 4M's we focus on are:

1. Molding process
2. Mold
3. Machine
4. Material

These 4M's are the key items that a troubleshooter can impact. The "man" is not included because a person can impact any of the 4M's. Each of the 4M's must be considered for potential root causes when troubleshooting. By reviewing the 4M's

it is much easier to troubleshoot with a systematic approach. By considering which of the 4M's could contribute and working through one category at a time a list of potential root causes can quickly be gathered.

All of the defects discussed in this book will use the 4M method for description of potential causes. Utilize the possible causes to systematically work through resolving the problem. Keep asking which of the 4M's could be contributing to the defect and why. Always try to drive deeper to get to the root cause of the problem. An example of using the 4M's is when troubleshooting sink: the natural place to start is with second-stage pressure; however, if the pressure is raised to compensate for a machine problem, was the true issue resolved or are you processing around another issue? The goal of the 4M method is to avoid processing around issues. Often times molders are left trying to work "process magic" to get good parts when a tooling improvement should have been implemented. Using the 4M method helps to keep process windows as wide as possible and will lead to less scrap, waste, and PPM (defective parts per million) in the long run.

Most people are familiar with the "5 Why" approach that was developed at Toyota. This approach is a tool that systematically drives toward asking questions about the root cause. In this approach, the goal is to get to the true root cause by asking why after every answer when problem solving. Many people find this technique useful.

One key to a systematic approach to troubleshooting is to review what has possibly changed in the mold, molding process, material, or machine. Frequently people will work on trying to fix a problem but not address what had actually changed that originally led to the problem. In other words, sometimes technicians are struggling to solve the wrong problem. A common example of this is someone slowing first-stage velocity to fix a burn that was actually caused by dirty mold vents. Using a systematic approach will help to focus on the true root cause of the problem and not to process around an issue.

The mentality to keep when troubleshooting should be to try to remove one potential root cause at a time. Until an issue has been proven to have no effect it remains a potential root cause. Using a systematic approach allows a troubleshooter to remove one cause at a time, focusing initially on the most likely causes and working from there. Always remember though that data is key to proving a root cause.

Change one thing at a time and determine the impact. If a troubleshooter changes multiple things at a time it is impossible to determine what the root cause was. After making a change, always give the molding machine time to stabilize before evaluating the impact of the change. If the process change shows no impact on the defect, it can be reset to the original documented process.

It is also vital to make changes that are large enough to have a potential impact. Frequently processors will make an adjustment to a process and when they do not

see an impact they scratch that variable off the list of potential causes. Remember that if the change is too large and causes other concerns it can be adjusted back towards the original setting. Make sure a parameter has been thoroughly evaluated before it is removed as a potential root cause.

1.4.2 STOP: Think

Think is the step to make sure that a troubleshooter has mentally reviewed the defect and the potential causes that were systematically determined. Before making a change, it is critical to think through what the expected result is as well as potential side effects. Always begin the think step with the question of "is this a new problem or has it been ongoing?" If it is a new problem focus on what changed; with an ongoing problem the focus is more on what needs to be corrected.

Sometimes in the think step of troubleshooting it is necessary to think outside of the box. Many problems encountered in molding are not easily solved and may require a creative approach to resolve. Willingness to not be constrained by comments such as "that's not the way we do it" is key to resolving problems. As Albert Einstein said, "we cannot solve our problems with the same thinking we used when we created them." There are many examples of molds where someone said that an area cannot be vented or cooled but through some ingenuity a solution was found. Remember that there are many exceptions to the general "rules of thumb"; critical thinking is vital.

Also, when thinking through a problem, think bigger than the current defect that is in front of you. Always ask if this problem may be happening elsewhere but has not been detected there. In the case of the 4M machine category, any mold that runs in that particular machine may be having problems but some will be worse than others. If one drying hopper is feeding multiple machines a splay problem may start to show up in multiple parts. Think about the root cause and what else it may impact and examine other parts that could be experiencing similar problems.

When thinking about a problem look for opportunities to push the thought process as far up front as possible. Effort put into part and mold design will result in improved process windows, reduced scrap, and more efficient launches. It is much more cost effective to ensure that the initial design is suitable for manufacturing rather than trying to correct mistakes after the mold has been built and run.

1.4.3 STOP: Observe

Observation is critical to solving problems. Much like Sherlock Holmes, a good molding troubleshooter must observe as much as they can regarding the problem and environment.

Observation should be a multiple sense process, meaning look, listen, and even smell what is happening at the molding machine. Visual examination of the parts, the equipment, and the process will most often provide valuable clues. However, when observing a molding machine in operation, the smell of degraded plastic may be an overwhelming indicator of a problem. Strange noises can also be an indication of something wrong in the process. Always observe with all senses to try to discover any clues to the cause.

When observing a molding process, a walk around the machine is usually a good practice. A quick walk can often highlight a concern that must be addressed. Key things to look for include:

- Auxiliary setpoints and actual values
  - Hot runner controllers
  - Thermolator
  - Chiller
  - Dryer
  - Gas assist equipment
- Clamp and robot movements
- Trimming operations
- Operator handling
- Material identified and correct
- Clear standards available?
- Anything that is damaged or out of place

Figure 1.1 shows a simple chart called the 4M Basic 8. These are the basic items that need to be observed during initial troubleshooting. Many problems can be resolved by simply working through these eight questions, and a “no” answer for any of these questions indicates a likely starting point for resolving the problem. The 4M Basic 8 is a very simple procedure that all molders should be able to work through and answer prior to calling for technical support. Utilizing the 4M Basic 8 or something similar as a starting point for troubleshooting puts good habits in place for troubleshooters.

	4M Basic 8	Y/N
	Molding Process	
1	Match documented setup?	
2	Fill Only Weight?	
	Mold	
3	Vents clean and open?	
4	Mold not damaged?	
	Machine	
5	Same machine?	
6	Machine achieving setpoints?	
	Material	
7	Correct material?	
8	Dry?	

Figure 1.1 4M Basic 8

Another key to the observation step of the STOP methodology is to ensure that good baseline data is available. Scrap reports are a critical piece of data to determine what the baseline defect rate is. Figure 1.2 shows a pie chart that provides a breakdown of the key scrap items for a particular job. Based on the Pareto Principle a likely expectation is that 80% of the scrap is a result of 20% of the potential root causes. This pie chart provides an easy reference tool to determine where the troubleshooting efforts should be focused.

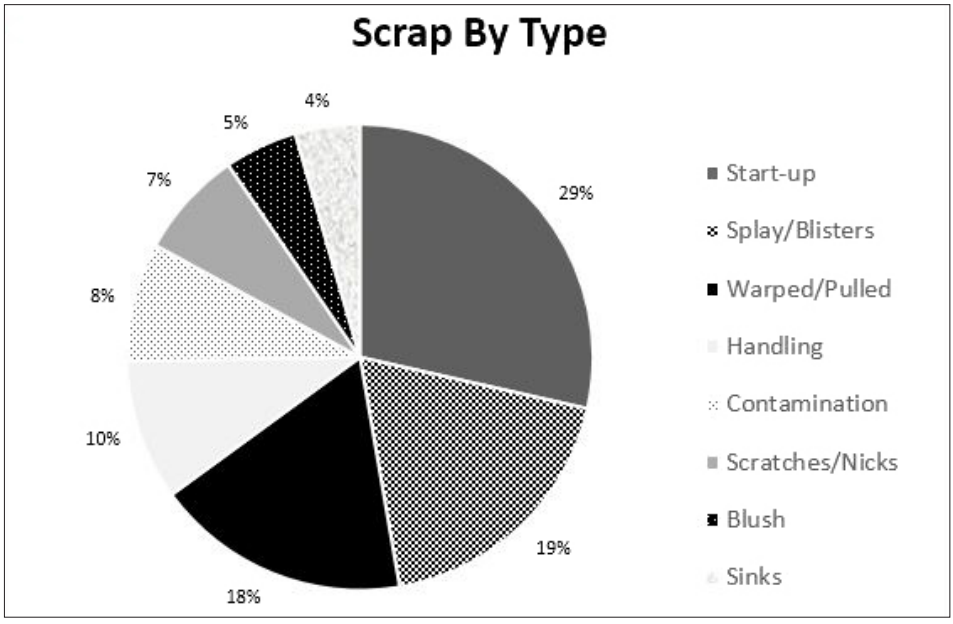


Figure 1.2 Pie chart breakdown of scrap percentage

A key observation task when reviewing data during troubleshooting is to evaluate if the problem has been an ongoing issue or has just recently started to occur. Figure 1.3 shows a graph that greatly illustrates an example of a sudden appearance of a defect. The part had been running with very little contamination scrap (less than 10% of total scrap) but then in June the contamination scrap numbers started to rapidly increase. The job continued to run poorly for approximately 5 months until the root cause was determined (problem with agglomeration of colorant components in the color concentrate). Validation of the improvement was simple due to the rapid drop of scrap in November.

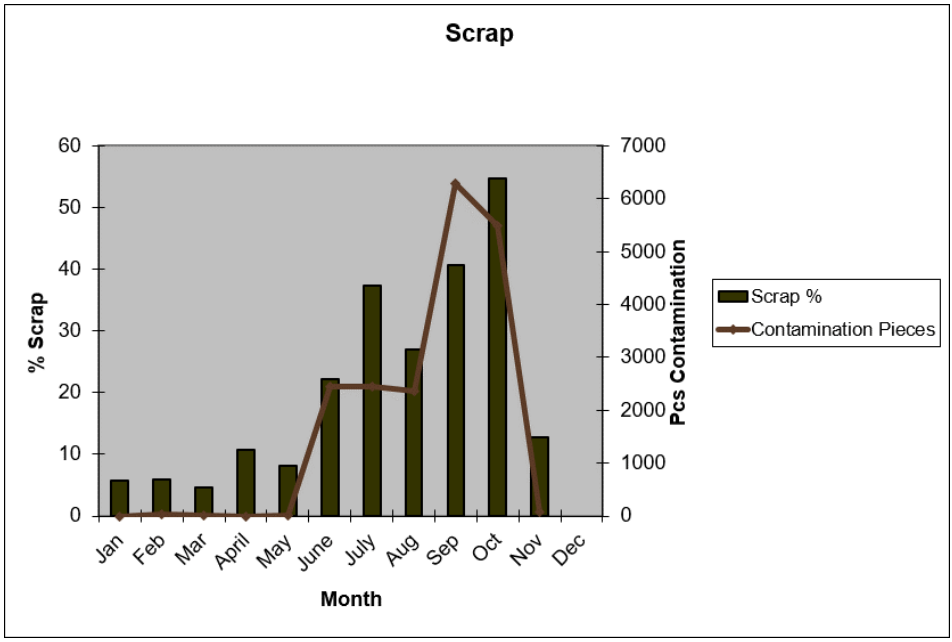


Figure 1.3 Graph showing a sudden increase in scrap and a corresponding sudden drop off in scrap after the problem was fixed

If a problem suddenly occurs the most important question to answer is "what has changed?" The power of observation is critical to determining what potentially changed. The 4M Basic 8 helps to evaluate possible changes and this simple step should always be done before diving deeper into the problem-solving process. It is important to understand that a sudden change may not have been something that someone did intentionally. Things that must be observed for possible unintentional change include:

- Shop environment
- Material variation

- Machine wear or damage
- Mold wear or damage

Any change in the above factors may lead to a condition that exceeds the ability of the molding process for producing quality parts.

When observation of the baseline data shows that the problem has always existed, the main question is not "what changed?" but rather "what must be changed?" There are many parts in manufacturing operations that have such small process windows that they produce a steady stream of scrap. If the part has been steadily producing scrap it is often more difficult to troubleshoot because the problem may be rooted in several factors and not in a simple change. If observation shows that the part has had defects since it was launched, all of the 4M's must be reviewed in depth for possible causes. A common situation found in troubleshooting is where a process was established to work around another issue such as venting in a mold. Rather than finding and correcting the root cause during process development, the processor worked his/her magic and developed a process that produced "good" parts. Sometimes it is only after the mold has had some run time in manufacturing that the true ramifications of the narrowed process window are understood.

In many cases a troubleshooter will find that ongoing scrap problems are rooted in processing around a mold, machine, or material concern. It cannot be stated enough that molders must not process around problems but rather need to have the problems fixed to maximize process windows and minimize scrap and cost. Putting "Band-Aids" on a problem will not help establish a robust process; fix the problem! To effectively resolve problems the technical groups must work together. If the maintenance or tooling department will not fix the problems that are encountered the processor is left holding the bag and will have to process around a root cause.

1.4.4 STOP: Proceed

This is the step that everyone is anxious to get to because this is where actual changes are tried. The problem that frequently occurs is that people will jump to trying things without going through the systematic, thinking, and observing phases of troubleshooting.

Jumping right into making changes can lead to damage to equipment or molds. Figure 1.4 illustrates an example from someone who jumped to a solution. In this case, the machine was producing short shots and losing cushion, so the technician increased the shot size to add more plastic to the cavity during first-stage inject. The problem here was not a shot size issue but rather where the plastic was going. As the photo shows the hot runner manifold was leaking and was encapsulated in

plastic while running. This hot runner manifold had to be torn apart and cleaned, have the leak repaired, and be rewired with new heaters and thermocouples.

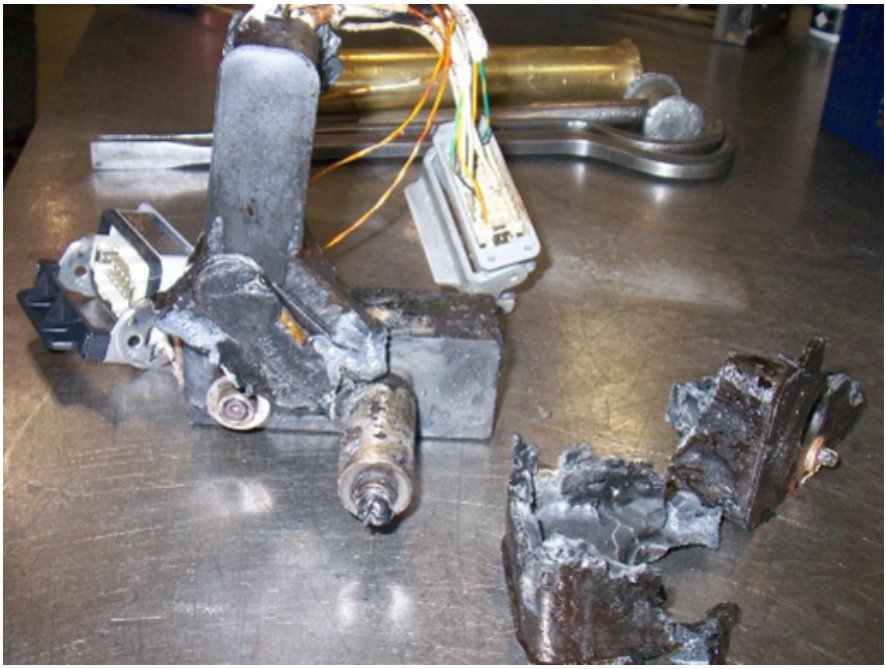


Figure 1.4 Effect of jumping to action without understanding the problem: a hot runner manifold encased in plastic

When the systematic, think, and observe steps have been completed and a direction is determined, it is time to make the change and evaluate the impact. Keep in mind that when the problem gets worse, that is indicating that the setpoint may have been adjusted the wrong way. Change is key when observing the impact of an adjustment: if there is no change the parameter is not the root cause.

When an adjustment is made that seems to improve the defect evaluate the following:

- Is the process adjustment within the allowed process window?
- Is the part cosmetically acceptable?
- Is the part dimensionally acceptable?
- Does the part meet all testing requirements?
- Does the part meet all other requirements?

If an adjustment successfully solves a problem make sure to dig a step deeper:

- Why was this adjustment required?
- What impact did the adjustment have on the plastic conditions in the cavity?

- Is the new condition stable and capable of producing quality parts for the long term?
- Is there anything else that should be done to ensure an adequate process window?
- Could this problem be affecting other product being produced?
- Does something in the company systems need to be changed to prevent this problem from reoccurring?

During the proceed step of STOP troubleshooting it is important to remember to review the results. If a process technician or engineer makes a change and does not examine the results they will have no way of evaluating if the change made a difference. There have been plenty of cases where a change was made and everything seemed good only for it later to be found that the data showed no difference. Always compare the results from after the change with the baseline data to determine if the change truly made a difference.

Some problem jobs may have a proceed step that lasts for an extended period of time to monitor the results to ensure that the problem has been eliminated. When all is said and done the data will tell the story.

1.4.5 STOP: Troubleshooting Cycle

It is critical to understand that the STOP methodology should be used as a cycle. After proceeding to make change it is time to go back to Systematic, Think, and Observe to work through evaluating if the change had the desired impact and if not, why? See Figure 1.5 for an illustration of the STOP troubleshooting cycle.

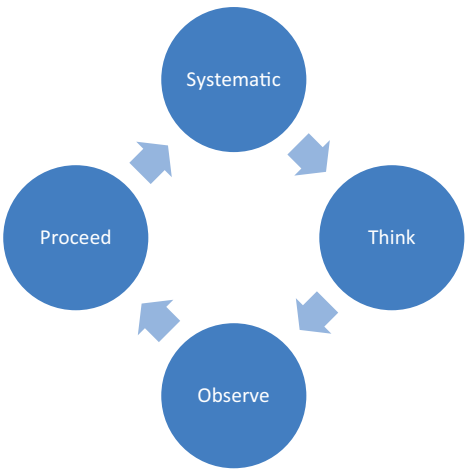


Figure 1.5 STOP Troubleshooting cycle

Continue to work through the STOP methodology cycle until the root cause is determined, corrected, and verified. Remember that it may take weeks of run time to fully determine the true impact of the change.

1.4.6 Hard Fix versus Processing around Problem

Whenever possible a permanent hard fix should be implemented to resolve the true root cause of the issue. If a physical change can be implemented in the mold or part design, the process window tends to be much larger. Implementing a hard fix to a mold may involve some up-front cost but will lead to long-term savings. Do not be short-sighted when troubleshooting, but rather think of the multiple years that the mold may run in production.

The importance of implementing a hard fix becomes more magnified when considered against the human ability to detect part defects. If the plant quality system is relying on detection from a person to catch defects the resulting customer returns will become very costly. However, if a hard fix is implemented the defects are prevented and human error and judgment do not play into shipping bad parts to the customer. Do not rely on human detection; prevention is the key to eliminating shipping bad product.

1.4.7 Troubleshooting Tools

In addition to the 4M methodology discussed above there are many additional tools and techniques that can help with troubleshooting including the following:

- 5 Why
- Fishbone diagram
- Scrap recording sheets
- Brainstorming
- Design of experiments (DOE)
- Is/Is Not
- Change log

1.4.7.1 5 Why

5 Why is a process that was developed by Sakichi Toyoda and used initially at Toyota for problem solving. The intent of the 5 Why process is to continue to ask questions until the true root cause is determined. An example would be:

- Problem with splay on parts.
  - Why? Material is wet.

- Why? Lack of drying time.
- Why? Hopper was not loaded in time.
- Why? Machine did not have an auto loader and had to rely on a material handler to load the dryer.
- Why? A decision was made to not spend the money to purchase the auto loader.

Using the 5 Why process helps drive down to the ultimate root cause by continuing to dig for the root cause. There will be times when the root cause is determined before 5 Whys have been asked and in some cases, it may even be required for additional questions to be asked.

1.4.7.2 Fishbone Diagram

The fishbone or Ishikawa diagram was developed by Kaoru Ishikawa in the 1960s. The diagram is frequently called a fishbone because of the fish-like shape that the diagram is drawn as. A fishbone typically looks at man, method, machine, measurement, and materials as categories of potential root causes. Each of the five categories is placed on the top line of the fishbone diagram and then additional bones (lines) are added below it to detail potential root causes.

As the 4M troubleshooting methodology was developed it became clear that the key categories molders need to worry about are molding process, mold, machine, and material. As mentioned previously the additional M of the man/woman can impact any of the 4M's.

1.4.7.3 Scrap Recording Sheets

One of the keys to effective troubleshooting is to have good data. There are many types of sheets that can be put together to capture the required scrap data. At the basic level, having a Pareto chart of the various defects is critical.

Another useful tool may be a tally sheet that documents scrap by the hour throughout the day. This can help to determine if a defect if it occurs at specific times or more during specific shifts. Sometimes these tally sheets may show that at the start of each shift the scrap increases until the operator gets up to pace. See Figure 1.6 for an example of a scrap tally sheet; notice there is a box for each hour of the shift for specific defects.

An additional scrap tracking sheet that can be useful is a printed-out example of the part. The operator than simply makes a mark where the defect occurred on the part. Using this type of tally sheet will help establish if a defect is in a consistent location or scattered throughout the part. In some cases, leaving a part at the machine to mark the defect locations can also work.

Part # _____ Initials _____
Date _____
Shift

	1	2	3	4	5	6	7	8
Blush								
Splay								

Figure 1.6 Scrap tally sheet

1.4.7.4 Brainstorming

Brainstorming can be conducted in a variety of fashions but is basically intended to gather some people together to capture as many potential causes as possible. The normal intent of brainstorming is that people can feed off of the ideas of others to build on concepts. Brainstorming can be an effective way of establishing potential root causes.

Some of the common brainstorming techniques include:

- Wide open. In wide-open brainstorming everyone can call out ideas as they come to them. The advantage of this method is that people may build on another's ideas. The disadvantages of this method are that quiet people might not speak up, and the free-flowing dialogue may disrupt creative thinking (or help it depending on the person).
- Round robin. In the round-robin approach a facilitator asks people around the room for an idea and then documents the idea. Advantages of this method are that quiet people get a chance to contribute and it is more focused than the wide-open method. A disadvantage is that there may be not as much of a group dynamic.
- Silent start. In this method everyone starts out the session by writing down a list of ideas. After a set period of time the facilitator asks for ideas and then documents them. Advantages of this method are that it limits group-think concerns and may allow more focused thinking. Disadvantages can include less initial discussion and leapfrogging from others' ideas.

Another thing to think about is adding both subject area "experts" as well as outsiders. Often times the outsiders will ask questions that can spark non-traditional thinking which in turn may lead to the true solution.

Alex Osborn is known as the originator of brainstorming and the basic rules he established are:

1. Do not criticize.
2. Wild ideas are welcome.
3. Go for quantity.
4. Build on others' ideas.

1.4.7.5 Design of Experiments (DOE)

DOE is a tool that creates an intentional set of experimental conditions that will help determine the impact of problems. The power of DOE is that it combines conditions in a way so as to limit the total number of experiments conducted during a trial. By utilizing DOE it is possible to capture interactions between process conditions and see what the major contributing factors are. There are numerous software packages on the market that can help develop DOE and analyze the data. Specifics on DOE are beyond the scope of this book, but it should be considered as a troubleshooting tool.

Sometimes simple full factorial DOE is the ticket to finding a solution. In a full factorial DOE there is no simplification of the number of experiments to be run. An example of a full factorial DOE may be looking at the influence of mold temperature and second-stage pressure on part dimensions. A full factorial DOE in this case would require the following runs:

1. Cold mold, low second-stage pressure
2. Warm mold, low second-stage pressure
3. Cold mold, high second-stage pressure
4. Warm mold, high second-stage pressure

These four experiments will determine the impact of these two factors. Adding a third factor would increase the number of experiments resulting in more time required to conduct the trials.

1.4.7.6 Is/Is Not

Is/is not can be applied as a simple tool to help narrow the scope of a problem. The way to conduct an is/is not evaluation is to make a chart with headings of "is" and "is not". The problem is then broken down into statements about what it is or is not, as shown in Figure 1.7.

Issue: Splay on mold #1234

Is	Is Not
Occurring only on mold #1234	Occurring on any other molds
Located randomly on part	Isolated to specific areas
Happening all day long	Happening at specific times

Figure 1.7 Is/Is Not example

1.4.7.7 Change Log

A change log can be used to help keep troubleshooting systematic by providing a way to track the changes made. A change log can be something such as Figure 1.8, which provides a simple sheet to record any changes and the impact that they had on the defect. This can be handy for communicating across shifts so everyone can see what was adjusted and the impact the change had on the problem.

Change	Better	Worse	Same
Increased fill time from 3.1 to 3.5 seconds		X	
Decreased fill time from 3.1 to 2.7 seconds	X		
Increased melt temperature 5°C			X
Decreased melt temperature 5°C			X

Figure 1.8 Example of a change log

1.4.8 Troubleshooting Methodology Summary

There are many useful tools that can be applied to troubleshooting. This chapter detailed many of these tools but especially focused on the STOP methodology of troubleshooting and the 4M method.

STOP troubleshooting focuses on:

- Systematic
- Think
- Observe
- Proceed

The 4M method is how all of the defects in this book are discussed. All defects are considered based on:

- Molding process
- Mold
- Machine
- Material

Be sure to ask whether the problem is a new issue or has been ongoing. This can help focus the troubleshooting because new problems need to be approached by looking for what changed.

Troubleshooting is a skill that can be built with knowledge and experience. Be willing to ask questions and dig for answers. Change one thing at a time and allow the process to stabilize.

Make sure that any changes are evaluated for unintended consequences. Be careful not to trade one problem or defect for another. If an adjustment helps reduce scrap but causes assembly problems, the end result may cost more in the long run.

2 Troubleshooting Tool Kit

There are many useful tools that should be available for troubleshooting in a molding operation. Without having critical troubleshooting tools, it will be nearly impossible to effectively troubleshoot a molding problem. Some tools are costly and a plant may have only one available but basic items should be accessible to everyone.

2.1 Lockout/Tagout

Whenever doing anything that involves working between the mold halves, it is critical to ensure proper lockout/tagout of the equipment.

2.2 Hand Tools

A variety of basic hand tools should be on hand when troubleshooting. These tools may include:

- Wrenches, including Allen, adjustable, and box end
- Screwdrivers
- Brass rods, scrapers, and brushes
- Bronze or brass pliers
- Slide hammer
- Various other tools as needed

2.3 Pyrometer

The melt temperature cannot be judged by reading barrel temperatures nor can it be judged by reading a thermolator. For determining actual temperatures, a pyrometer with a fast-acting melt probe and a contact probe must be available. Actual temperature readings are critical when troubleshooting!

2.4 Spotting Blue

To check the surface contact of mold surfaces spotting blue is used. It should be brushed on one surface of the tooling and then the mold is closed and opened and then inspected for transfer of blue onto the clean surface. This is used to verify shut offs and if vents are open.

2.5 Measurement Tools

It is very handy to have a variety of measurement tools on hand. Calipers and micrometers will help give wall stock measurements. Often times a tape measure is more fitting for examining larger dimensions such as size of mold versus platens, total flow length, etc. A depth micrometer is useful for checking actual vent depth.

2.6 Multimeter

In trained hands, a multimeter can be a useful tool for troubleshooting. Whether verifying volts to temperature for thermocouple or checking continuity on a hot runner cable, there are things that need to be checked with a multimeter.

2.7 Process Monitoring Equipment

Process monitoring equipment like RJG's eDART® can be very useful for diagnosing problems at a deeper level. The biggest advantages to using a process monitoring system to help troubleshoot are as follows:

- Allows collection of large amounts of data including:
  - Machine pressures
  - Cavity pressure
  - Mold temperatures
  - Cooling
  - Fill speed
  - Fill profile
  - Dryer function
  - Weather
- Allows storage of data for easy analysis
- Allows a baseline template to be saved
- Allows capture of intermittent problems; it captures data on every shot

The use of process monitoring equipment can be a tremendous benefit to the troubleshooter. It can provide all the data detail to dive deep into every phase of the process.

2.8 Moisture Analyzer

A moisture analyzer is a useful tool to help determine if the material has been adequately dried. A loss on weight moisture analyzer can give a false high reading, however, because it will measure the total weight loss during heating of the material, which could include volatiles other than water.

2.9 Dew Point Meter

To verify that a dryer is working properly the dew point must be measurable. Some dryers have built in dew point monitors but portable meters can be purchased from industrial supply shops. If material drying is the root cause of the problem, con-

ducting a dew point check on the dryer will determine if the dryer is capable of reaching adequate levels (typically −40 °F).

2.10 Flashlight

A flashlight is a cheap tool that is valuable to help see better detail on a part. An example is EDM marks or lack of polish on a core will be much more apparent when using a flashlight for inspection. When looking into a mold that is in the press, shadows will obstruct a clear view, but often times a simple pen light will solve this problem.

2.11 Microscope/Magnifying Glass

Sometimes magnification is the only way to identify what a defect really is. Having a magnifying glass and/or a portable microscope can allow a much closer examination of the defect to determine for example if it is splay or a scratch. Portable USB microscopes are now available for low cost and allow photographing of a defect or area of concern, which is very useful when communicating a problem.

2.12 Silly Putty

Silly putty is a very simple item that can prove useful. When wondering if a particular defect is in the mold steel, pressing silly putty unto the area can sometimes provide a clear answer. It is also very useful for looking at small and hard-to-see details such as date wheels.

2.13 Inspection Mirror

Inspection mirrors are a handy item to help see hidden areas in a mold, and can as well as be used to inspect other challenging areas such as feed throats.

2.14 Thermal Imaging Camera

Costs of infrared thermal imaging cameras has come down to a level that is much easier to justify. Use of a thermal imaging camera on parts will help provide an accurate view of the "part out temperature" after the part is ejected from the mold. These thermal images can be used to detect hot spots that may indicate cooling issues. Thermal imaging of nozzle and barrel heaters can also indicate hot or cold spots that should be addressed. Thermal imaging is a very useful tool to help with cycle time optimization; in fact, the savings from this alone will justify the capital investment in a camera.

2.15 Aluminum Tape

A quick check to determine if improved venting will solve a problem is to add aluminum tape to the parting line near the area of concern to allow for additional venting. If this improves the issue the venting should be added or deepened.

See also Chapter 7, which covers venting.

2.16 Dial Indicator

A dial indicator on an adjustable arm can be used to check for mold deflection. When troubleshooting flash, it is critical to know if the mold is being blown open with injection pressure. Setting up a dial indicator and watching for movement can usually prove if clamp tonnage is a root cause.

2.17 Purging Compound

When dealing with contamination-type defects, having a purging compound can allow the barrel and machine to be cleaned out of contaminating resin and carbon buildup (at least to a point).

2.18 Grinder/Stones

Access to grinding tools and stones will allow vents to be opened up on a mold suffering from venting related problems. Caution with these items is extremely important so as not to cause damage to the parting line of the mold, which would in turn lead to flash.

2.19 Camera

With cell phone cameras, this is an easy item to have on hand. Using a camera to document defects and impact of changes will allow a clear way to communicate the problem and review any changes.

2.20 Material Data

When troubleshooting it is key to have the material molding recommendations available. This provides an opportunity to cross reference important process values such as melt temperature, mold temperature, drying parameters, venting, and gate recommendations as well as many other process details. Make sure that this data is readily accessible for all troubleshooters.

2.21 Scale

A scale is critical for accurately measuring part weight. Without an accurate scale it is not possible to replicate a fill only shot, which is one of the key checks for a molding process. Also having the ability to measure shot weight will allow analysis of dryer throughput, and approximate barrel residence time.

2.22 Flow Meter

A flow meter is used to measure water flow in a mold. Through the use of a flow meter it is possible to determine if a water line has adequate flow to achieve optimal cooling. Checking the flow across water circuits may show change over time that can indicate problems that can impact the ability of the mold to cool the plastic.

2.23 Mold Cleaning Supplies

A good first step for troubleshooting is to clean the mold using mold cleaner and a wipe. Many defects can be impacted by a dirty mold. Before trying process adjustments, the mold should be cleaned. If the defects go away two things may need to happen:

1. Create a cleaning standard for how and when. Every shift should be cleaning the mold appropriately and this should be occurring on a regular basis. Also standardize cleaning at mold start ups.
2. Evaluate the mold for adequate venting. If the process technicians cannot keep up with cleaning the mold it most likely does not have enough venting.

More effective cleaners such as Zapox may be required to clean significant buildup from a mold surface.

2.24 Miscellaneous Supplies

There are a number of other simple items that make troubleshooting easier including:

- Note pads to help keep track of changes
- Markers to write on sample parts
- Ziplock bags for bagging samples
- A calculator to help with math
- A sealed container for samples for moisture analysis (i.e., glass bottle)

3 Decoupled® or Scientific Molding

Most of the methods discussed in this chapter are those developed and taught by Don Paulson [1], John Bozzelli [2], and Rod Groleau [3]. There is a large amount of excellent information available on Decoupled®/scientific molding, so this chapter will only cover the basics [4].

One of the Merriam-Webster dictionary definitions of scientific is as follows:

Conducted in the manner of science or according to results of investigation by science: practicing or using **thorough or systematic methods**.

This is the foundation that Decoupled®/scientific molding is built on, thorough and systematic, and is also a great definition for how effective troubleshooting should be conducted.

The basic premise of Decoupled®/scientific molding follows these steps:

1. Fill the mold to 95–98% full using first-stage velocity control. Filling should be as fast as possible while making a quality part. The molding machine must not be pressure limited meaning that the required pressure to achieve desired fill velocity should not reach the machine's maximum pressure (actually anything within 2000 psi plastic pressure should be considered pressure limited; this allows for compensation due to viscosity variation). The 95–98% short shot is referred to as a fill only shot.
2. Conduct a cavity balance study if running a multi cavity mold. See Chapter 12 (Cavity Balance) for more details.
3. Transfer from first-stage velocity control to second-stage pressure control by using the transfer position setting on the machine.
4. Control pressure during second stage to compensate for plastic shrinkage during cooling.
5. Second-stage pressure should be controlled for a set time period. This time period should be determined by conducting a gate seal study [5]. For every mold determine if running with gate seal provides the best quality part.
6. Cooling time will be optimized to minimize cycle time, based on part ejection temperatures.

7. The screw should normally recover to shot size approximately 2–3 seconds prior to mold open. This may require use of a screw rotate delay to avoid rotating the screw at extremely slow speeds.

These basic steps are the process methods that this book will reference. Note that the particular process method is Decoupled II® molding as defined by RJG Inc.

The key is to use a scientific approach and let data help with decision making. Decisions should be made based on data. Use of the STOP method (see Chapter 1) will help reinforce making the effort to gather the data. Also when using this method, focus on recording the plastic data; for example, record fill time and fill only weight rather than worrying about velocity set points. By using plastic data it is possible to translate from machine to machine and process to process.

Modern scientific molding relies on establishing and documenting a process and then always running that process. The days of process technicians having a "little black book" of their personal processes are long gone. If someone has an idea to improve a process it must be approached with data, proper evaluation, and validation. The process technicians should understand that even though they think the process is better all impacts must be examined. There is a saying that the "law of unintended consequences" can create wide-ranging impacts that were never considered. Depending on customer and part requirements it may be necessary to resubmit parts and extensive test data to support a process change.

From a practical standpoint everyone in the molding operation must be knowledgeable in scientific molding methods. Once people understand that the process was based on decisions that are backed with data they will tend to take more ownership of the process variables. Quality training in the techniques is absolutely critical to success.

Benefits of Decoupled®/scientific molding include:

1. Faster fill rates yield faster cycle times.
2. Faster fill rates yield lower and more consistent viscosity. Lower viscosity allows easier filling of the mold and less pressure drop across the cavity.
3. Separating first-stage fill from second-stage pack/hold gives the processor the ability to impact plastic flow rates and plastic pressurization as independent points of control.
4. Utilizing a fill only weight of 95–98% will help prevent mold damage and flash from rapid spikes in cavity pressure.
5. Use of gate seal studies ensures that second-stage time is not wasted packing out runners.
6. Cavity balance studies ensure that each cavity is experiencing the same plastic conditions throughout the process.

7. Data generated during process development is always available for later trouble-shooting.

Utilize computer-based spreadsheets to create process "setup" forms that capture the data generated during molding process development. With some creativity a spreadsheet can easily be constructed that provides all of the calculations and graphing that is needed to define a process. Selection menus can be created that allow selection of machines or materials that automatically populate specific portions of the spreadsheet, for example intensification ratio and tonnage for a desired machine. See Figure 3.1 below for an example of a cavity balance spreadsheet form.

Cavity	Cavity weight	Balance %
1	112	97.4
2	114	99.1
3	114	99.1
4	115	100.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0

Mold % Imbalance: 2.6

((Highest weight - lowest weight)/(highest weight)) X 100

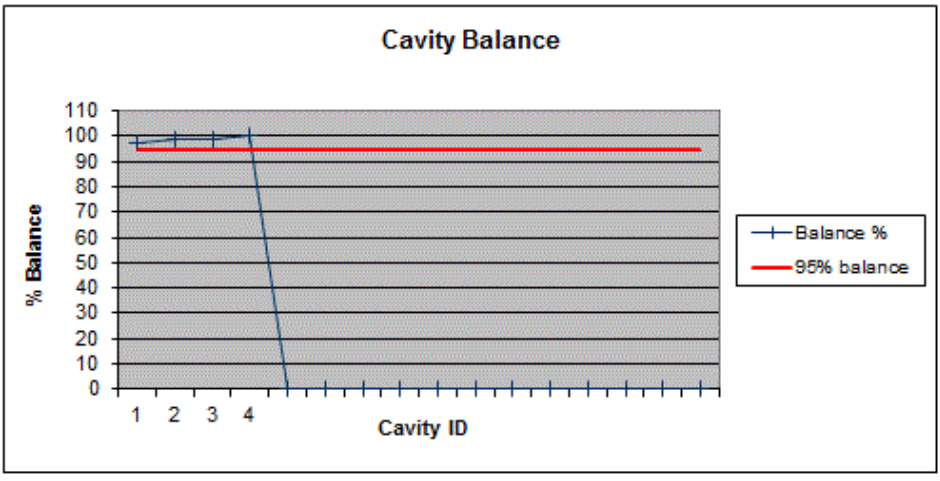


Figure 3.1 Cavity balance spreadsheet

Whenever possible a process should be kept as simple as possible. If first-stage fill does not need to be profiled why use six steps of profiling? Every extra level of complexity in the process leads to more opportunities for a mistake to happen. It is much easier to validate that the fill only weight and fill match the documented process than to try to determine if every step of a complex profile matches what it should. As Albert Einstein put it, "everything should be made as simple as possible, but not simpler."

To be successful with these molding principles it is absolutely critical to not process around material, mold, or machine problems. The 4M method of molding troubleshooting (see Chapter 1) helps to focus on eliminating problems. Instead of blaming the process because fast fill rates lead to burns, it is much better to correct the venting on the mold than use the process method as an excuse. Scientific/Decoupled® molding is very effective but molders cannot expect success if they are trying to process around other issues.

Another key to successful implementation of scientific/Decoupled® molding methods is that all processors in the molding plant must understand the tools used to develop processes and be confident that they work. If a process is developed in a typical Decoupled II® methodology with a fill only weight at 98% full and someone decides to fill the part farther it is likely that the mold will flash. Blaming this flash on the method is foolish; instead the processor must understand the tools that the process is built with. Training is foundational to successful molding, as without adequate training the personnel will feel like the processes are being "tossed over the wall" into manufacturing and may not provide the support required to supply quality parts. Incorporate thorough training for all of the process engineers, technicians, die setters, and material handlers to help eliminate learning from the "school of hard knocks".

References

[1] Paulson Training Programs Inc., 3 Inspiration Lane, Chester CT, 06412; Tel. (860)526-3099; https://www.paulsontraining.com/

[2] Injection Molding Solutions, 1019 Balfour St., Midland MI, 48640; Tel. (989)832-2424; http://www.scientificmolding.com

[3] RJG Inc, 3111 Park Dr., Traverse City MI, 49686; Tel. (231)947-3111; https://rjginc.com/

[4] Groleau, Rod, "The Fundamentals of Decoupled Molding", *Plastics Today*, May 2005

[5] Bozzelli, John, "Why and How to Do Gate Seal Experiments", *Plastics Technology*, Oct 2010

4 Gating Details

4.1 Gating

Gating is a subject that has not had extensive research and most do not understand the impact it has on the process. The gate or hot-drop orifice in most cases is the most restrictive point in the flow of the plastic and so it should be. Gating can impact gate seal time, pressures, cavity imbalance, and defects such as high gates, flaking, jetting, and blush. And when gate modifications are made it is typically an increase in size or gate volume and not the opposite with a decrease in size or volume. But to address some of these issues a reduction in gate size or thickness can be the best approach and varies with the type of plastic you are molding. It is important to understand exactly what the purpose of the gate is, and there is not just one standard or one rule when it comes to gate standards with all the variables involved with part volume, flow lengths, wall stock, and the plastic itself. Always use the STOP process and make sure to think through what contribution the gate has on the process and defects.

Just because you decrease the gate size or thickness, it does not always mean that plastic pressure will increase. And increasing a gate thickness or size does not always mean your plastic pressure will drop. We mention this because it goes against common assumptions.

Most people automatically assume you will have an increase in fill pressures if you go thinner in gate thickness. We have found that it is more about volume, which can be maintained by increasing the width when reducing the gate thickness. Some materials prefer the higher shear with thinner gates. On the other hand, glass-filled materials with higher glass content need runners, hot-drop orifices, and gates as large as possible to reduce pressures and be able to pack out the glass.

Typically, when a gate is made smaller you can expect to see a change in plastic pressure because you have reduced the volume of the gate orifice. But a change in gate size or volume does not always produce a noticeable change in plastic pressure.

It is important not to focus on the gate as the only restriction to plastic flow. The gate land can be a contributor to pressure loss and is often overlooked. There is no advantage to excessive gate land and it can contribute to other issues also. Many people also overlook the role of hot-drop tip orifices. We have often seen people open up gates or runners when the hot-drop tip was the restriction—as would occur, for example, when a hot runner feeds into a cold runner. Remember it is about the area or volume of the hot-drop orifice or gate.

Plastics are understood to flow in what is referred to as fountain flow. Let's assume you have a round runner and experience textbook fountain flow. Then what happens when you take this volume of flow and ram it through a small round orifice into the large space of the cavity? But what if you could change the gate geometry to improve the transition from the runner to the part? Let's use a pressure washer as an example to help paint a mental picture. If you were to put a tip with small round orifice on your pressure washer, what type of stream would you expect to see? Now if you put a tip with a rectangular orifice that was thin and wide, what would the flow look like? The difference is pretty drastic between a small, straight, jetting stream and a fanned-out stream. With the thin-and-wide concept we have been able to reduce cycle times with a quicker gate seal, maintain or reduce fill pressures, eliminate high gates or vestiges on cashew gates, help minimize gate blush, eliminate jetting, eliminate pulls, and eliminate flaking.

Case Study

One example is a PC/ABS part where we were addressing high gate vestige. It had a cashew gate with a 0.040-in diameter orifice. We changed the orifice from 0.040 in round to a rectangular shape of 0.020 × 0.080 in, which increased the gate volume. In this case we were not only able to help reduce the high-gate defect but were able to drop the fill pressures from 16,000 to 11,000 psi. This created a larger process window on a part that had struggled with flash and shorts.

In another example we were able to eliminate two defects—jetting and pulls—on a larger glass-filled PP part that was causing a lot of scrap. The 750-ton tool had no process window allowance to address the defects. The part had two cashew gates with 0.110-in diameter round orifices. In this case we could go thinner but not wider because the geometry of the taper would not allow it. We started by welding up the orifice and going from the 0.110 in round to a 0.050 × 0.110 in rectangle. We were concerned about increasing fill pressures because we were reducing the orifice volume.

When we ran the tool after the change there was no increase in fill pressures. Jetting was improved but we still had some issues with the pulls. The pulls were the result of the part shrinking away from the gate area before ejection. So, we had another idea: If we made the gate thinner than 0.050 in, would it break while the part was shrinking away, reducing the pull

defect? Because the first change had no impact on fill pressures we thought it was worth a shot. We welded the gate orifice and reduced it from 0.050 × 0.110 to 0.025 × 0.110 in.

This time around our fill pressures did increase from around 10,000 psi to 13,000 psi but the pulls were eliminated and the process window drastically improved, with less jetting and thousands of dollars in scrap savings.

We worked on a program of PP parts that were having high-gate issues. In this case the cashew gates had a 0.040-in diameter. We went to 0.020 × 0.080 in, and the high gates were eliminated, and there was no increase in fill pressures.

4.2 Gate Size, Shape, and Taper

Figure 4.1 and Figure 4.2 show geometry alternatives to the industry standards. The D-gate style sub-gate versus the standard sub/tunnel gate simulates the thin/wide concept and will provide much cleaner gate breaks than the standard sub-gate. And with this style in one case we were able to reduce gate-seal time by 5 seconds, saving an equal amount in overall cycle time. With the D-style sub-gate the taper is taken out of the equation for flow restrictions because there is not a gradual taper of mass like the standard sub-gate. You can also sub-gate into angled walls, where when using a standard sub-gate you would end up with a gate vestige. Also, you can gate into shorter walls than with a standard sub-gate. We have gated into cavity walls that were 0.125 in tall. You have endless opportunities with D-gate orifices. You can increase the tip of the cone on the D-gate to create a wider orifice. It is important to understand the volume of the gate orifice. You can use a much thinner gate orifice because you are increasing volume with the extra width.

Cashew and Tab gates - Round/square versus elongated

Elongated advantages	Round/Square disadvantages
Reduced pressure loss/fill pressures based on area	Increased pressure loss/Fill pressures (when thickness same as elongated)
Reduced cycle time with gate seal	Increased cycle time with gate seal (With area same as elongated)
Less vestige with a cleaner gate break	Increases risk of vestige based on area
Reduces/eliminates high gate issues on cashew gates	Increases risk of high gates based on area
Reduces gate blush	Increased risk of gate blush
Reduces jetting issues	Increased risk of jetting
Better flow transition from the runner into the cavity	Does not always provide the best runner to part transition with flow

Gate Thickness	Tab/Square Gate	Round Orifice	Length/Thickness 2/1	3/1	4/1	5/1	6/1	7/1	8/1	9/1	10/1	11/1	12/1
0.010	0.0001	0.0001	0.0002	0.0003	0.0004	0.0005	0.0006	0.0007	0.0008	0.0009	0.0010	0.0011	0.0012
0.015	0.0002	0.0002	0.0004	0.0007	0.0009	0.0011	0.0013	0.0016	0.0018	0.0020	0.0022	0.0025	0.0028
0.020	0.0004	0.0003	0.0007	0.0011	0.0015	0.0020	0.0024	0.0028	0.0032	0.0036	0.0040	0.0044	0.0048
0.025	0.0006	0.0005	0.0011	0.0018	0.0024	0.0030	0.0036	0.0043	0.0049	0.0055	0.0061	0.0068	0.0074
0.030	0.0009	0.0007	0.0016	0.0025	0.0034	0.0043	0.0052	0.0061	0.0070	0.0079	0.0088	0.0097	0.0106
0.035	0.0012	0.0010	0.0022	0.0035	0.0047	0.0059	0.0071	0.0084	0.0096	0.0108	0.0120	0.0132	0.0145
0.040	0.0016	0.0013	0.0029	0.0045	0.0061	0.0077	0.0093	0.0109	0.0125	0.0141	0.0157	0.0173	0.0189
0.045	0.0020	0.0016	0.0036	0.0056	0.0076	0.0096	0.0116	0.0136	0.0156	0.0176	0.0196	0.0216	0.0236
0.050	0.0025	0.0020	0.0045	0.0070	0.0095	0.0120	0.0145	0.0170	0.0195	0.0220	0.0245	0.0270	0.2950
0.055	0.0030	0.0024	0.0054	0.0084	0.0114	0.0145	0.0175	0.0205	0.0235	0.0266	0.0296	0.0323	0.0356
0.060	0.0036	0.0028	0.0064	0.0100	0.0136	0.0172	0.0208	0.0244	0.0280	0.0316	0.0352	0.0388	0.0424
0.065	0.0042	0.0033	0.0075	0.0118	0.0160	0.0202	0.0244	0.0287	0.0329	0.0371	0.0413	0.0456	0.0498
0.070	0.0049	0.0038	0.0087	0.0136	0.0185	0.0234	0.0283	0.0332	0.0381	0.0430	0.0479	0.0528	0.0577
0.075	0.0056	0.0044	0.0100	0.0157	0.0213	0.0270	0.0326	0.0382	0.0438	0.0495	0.0551	0.0607	0.0663
0.080	0.0064	0.0050	0.0114	0.0178	0.0242	0.0306	0.0370	0.0434	0.0498	0.0562	0.0626	0.0690	0.0754
0.085	0.0072	0.0057	0.0129	0.0202	0.0274	0.0346	0.0418	0.0491	0.0563	0.0635	0.0707	0.0780	0.0852
0.090	0.0081	0.0064	0.0145	0.0226	0.0307	0.0388	0.0469	0.0550	0.0631	0.0712	0.0793	0.0874	0.0955
0.095	0.0090	0.0071	0.0161	0.0252	0.0342	0.0432	0.0522	0.0613	0.0703	0.0793	0.0883	0.0974	0.1064
0.100	0.0100	0.0079	0.0179	0.0279	0.0379	0.0479	0.0579	0.0679	0.0779	0.0879	0.0979	0.1079	0.1179
0.110	0.0121	0.0095	0.0216	0.0337	0.0458	0.0579	0.0700	0.0821	0.0942	0.1063	0.1184	0.1305	0.1426
0.120	0.0144	0.0113	0.0257	0.0401	0.0545	0.0689	0.0833	0.0977	0.1121	0.1265	0.1409	0.1553	0.1697
0.130	0.0169	0.0133	0.0302	0.0471	0.0640	0.0809	0.0978	0.1147	0.1316	0.1485	0.1654	0.1823	0.1992
0.140	0.0196	0.0154	0.0350	0.0546	0.0742	0.0938	0.1134	0.1330	0.1526	0.1722	0.1918	0.2114	0.2310
0.150	0.0225	0.0177	0.0402	0.0627	0.0852	0.1077	0.1302	0.1527	0.1752	0.1977	0.2202	0.2427	0.2652
0.160	0.0256	0.0201	0.0457	0.0713	0.0969	0.1225	0.1481	0.1737	0.1993	0.2249	0.2505	0.2761	0.3017
0.170	0.0289	0.0227	0.0516	0.0805	0.1094	0.1383	0.1672	0.1961	0.2250	0.2539	0.2828	0.3117	0.3406
0.180	0.0324	0.0254	0.0578	0.0902	0.1226	0.1550	0.1874	0.2198	0.2522	0.2846	0.3170	0.3494	0.3818
0.190	0.0361	0.0284	0.0645	0.1006	0.1367	0.1728	0.2089	0.2450	0.2811	0.3172	0.3533	0.3894	0.4255
0.200	0.0400	0.0314	0.0714	0.1114	0.1514	0.1914	0.2314	0.2714	0.3114	0.3514	0.3914	0.4314	0.4714

Figure 4.1 Chart comparing various gate areas

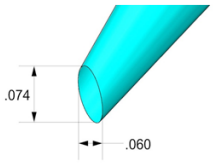
Case Study: Gate Differences

One example of this was on a two-cavity mold with sub-gates. This mold had a right-hand and left-hand part that were exactly mirrored so there was no difference between cavity size but the cavities were significantly unbalanced. I was asked to open up the sub-gate orifice on the cavity that was short even though the gate orifices were the same size (not always a good idea). After opening the orifice to where it had 50% more volume, I knew something was not right. I looked closely and found the sub-gate tapers were slightly different, but not enough to be noticeable without very close examination. I increased the taper on the cavity that was short to match the taper on the other cavity. Wow, what a change! The cavity that had been short now filled way ahead of the other cavity. A D-gate would have eliminated the imbalance because the taper is not gradual to the orifice.

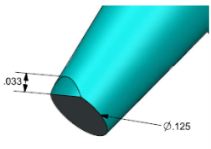
D-gate advantages compared to the standard sub-gate or tunnel gate.

D-Gate advantages
Reduced pressure loss thru the taper
Eliminates risk of cavity imbalance thru the taper
Can decrease cycle time with gate seal
Allows gating into tighter areas and radii
Reduced gate vestige
Reduced plastic flaking, cleaner break

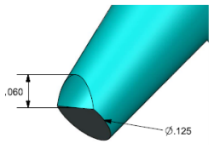
Sub-gate disadvantages
More pressure loss thru the taper
Can cause cavity imbalance if tapers are not exact
Can increase cycle time with gate seal
Increased distance in gate height/thickness
Increased risk of gate vestige
Increased risk of flaking



Standard .060 orifice sub gate
Gate area .0028 sq in



D-gate .033 thickness .125 cone
Gate area .0028 sq in



D-gate .060 thickness .125 cone
Gate area .0058 sq in

Surface area chart comparison. Standard sub-gate and D-gates

D-Gate Thickness	Standard Sub-gate	D-gate tip of cone diameter										
		0.032	0.046	0.062	0.093	0.125	0.156	0.187	0.250	0.312	0.375	0.500
0.010	0.0001	0.0002	0.0003	0.0004	0.0004	0.0005	0.0005	0.0006	0.0007	0.0007	0.0008	0.0009
0.015	0.0002	0.0004	0.0005	0.0006	0.0007	0.0008	0.0009	0.0010	0.0012	0.0013	0.0015	0.0017
0.020	0.0003	0.0005	0.0007	0.0009	0.0011	0.0013	0.0014	0.0016	0.0018	0.0021	0.0023	0.0026
0.025	0.0005	X	0.0009	0.0012	0.0015	0.0017	0.0020	0.0022	0.0026	0.0029	0.0032	0.0037
0.030	0.0007	X	X	0.0015	0.0019	0.0023	0.0026	0.0029	0.0033	0.0038	0.0041	0.0048
0.035	0.0010	X	X	0.0018	0.0023	0.0028	0.0032	0.0036	0.0042	0.0047	0.0052	0.0060
0.040	0.0013	X	X	X	0.0028	0.0034	0.0039	0.0043	0.0051	0.0057	0.0063	0.0074
0.045	0.0016	X	X	X	0.0033	0.0040	0.0046	0.0051	0.0060	0.0068	0.0075	0.0088
0.050	0.0020	X	X	X	0.0037	0.0046	0.0053	0.0059	0.0070	0.0079	0.0088	0.0102
0.055	0.0024	X	X	X	X	0.0052	0.0060	0.0068	0.0080	0.0091	0.0101	0.0118
0.060	0.0028	X	X	X	X	0.0058	0.0068	0.0076	0.0091	0.0103	0.0114	0.0133
0.065	0.0033	X	X	X	X	0.0064	0.0075	0.0085	0.0101	0.0115	0.0128	0.0150
0.070	0.0038	X	X	X	X	X	0.0083	0.0094	0.0113	0.0128	0.0142	0.0167
0.075	0.0044	X	X	X	X	X	0.0091	0.0103	0.0124	0.0141	0.0157	0.0185
0.080	0.0050	X	X	X	X	X	0.0099	0.0112	0.0135	0.0155	0.0172	0.0203
0.085	0.0057	X	X	X	X	X	X	0.0122	0.0147	0.0169	0.0188	0.0221
0.090	0.0064	X	X	X	X	X	X	0.0131	0.0159	0.0183	0.0204	0.0240
0.095	0.0071	X	X	X	X	X	X	0.0140	0.0171	0.0197	0.0220	0.0260
0.100	0.0079	X	X	X	X	X	X	X	0.0183	0.0211	0.0236	0.0280
0.110	0.0095	X	X	X	X	X	X	X	0.0208	0.0241	0.0270	0.0320
0.120	0.0113	X	X	X	X	X	X	X	0.0233	0.0271	0.0305	0.0362
0.130	0.0133	X	X	X	X	X	X	X	0.0258	0.0302	0.0340	0.0406
0.140	0.0154	X	X	X	X	X	X	X	X	0.0332	0.0376	0.0450
0.150	0.0177	X	X	X	X	X	X	X	X	0.0364	0.0413	0.0495
0.160	0.0201	X	X	X	X	X	X	X	X	0.0395	0.0449	0.0542
0.170	0.0227	X	X	X	X	X	X	X	X	X	0.0487	0.0589
0.180	0.0254	X	X	X	X	X	X	X	X	X	0.0524	0.0636
0.190	0.0284	X	X	X	X	X	X	X	X	X	0.0562	0.0685
0.200	0.0314	X	X	X	X	X	X	X	X	X	X	0.0733

Figure 4.2 Chart with D-gate area

Case Study: Thin Gate

We had a trailing ring gate on an ABS part with only a 0.006-in gate orifice. On a prototype I had no concerns with fill pressures using a 0.007-in thick gate. When we ran the new tool with a 0.006-in ring gate, our fill pressures were touching the limits on the high side. Needless to say, I was surprised because I had not seen this on the prototype, but the production tool was not an apples-to-apples comparison regarding cavitation or part geometry. We increased the gate thickness to 0.012 in, ran the tool, and the pressures did not drop at all. It was not the gate causing the high pressures.

Now think in reverse: If we had built the tool with a gate that was 0.012-in thick and the process was pressure limited, you would have never considered ▶

reducing the gate thickness to 0.006 in. You probably would have bet everything you owned that the pressures would have increased. A thickness reduction of 0.006 in may not seem like much, but it was a 50% reduction in gate volume.

Gating and its geometry can have a big impact on shear rates, gate seal, pressures, and quality issues. Most molders and mold makers have their own opinion on gate sizing and many do not keep an open mind on the impact gate size can have on the process and part quality.

To establish gate standards, you have to consider material variation, part wall thickness, and flow lengths. Typically, we would tend to err on the small side, going thin and wide when possible; but with some materials, such as glass-filled nylon, we tend to go much larger, especially with larger parts where there is more volume of plastic to move.

Most of the efforts in the industry to control the injection process involve measuring and controlling injection pressure. This is, of course, a very critical parameter to control and to confirm you have a quality part. But consider the measurement of actual injection volume through the gate, and how it relates to injection pressure, which possibly would be a great tool for process analysis. The gate is typically not a big focus unless there are pressure-loss issues. Pressure-drop studies are the common means to observe the impact of the gate on the process but the industry standard process is flawed and does not take into consideration the impact the gate has.

Case Study: Pressure Drop

A part and process was pressure limited at 24,000 psi, which was the maximum machine pressure available. This was a polypropylene part that had two gates and excessive flow length. The pressure-drop study showed that through the gates we had 9,000 psi pressure drop, and the rest was generated through the cavity. This specific grade of PP had a high viscosity, which contributed to the process being pressure limited. (Running a standard PP required a plastic pressure of only 9,000 psi.)

Some people would have suggested using a higher-pressure machine. I thought I could bring the pressure down slightly with a larger gate size but did not think I would be able to eliminate the pressure-limited issue. The gates were originally 0.020 × 0.080 in and I opened them up to 0.030 × 0.080 in. We then did a pressure-drop study through the new gates, and there was no change: it was still at 9,000 psi, which I found a little surprising. I then opened the gates to 0.040 × 0.080 in and the pressure-drop study still showed 9,000 psi pressure loss through the gates.

Looking back, I see it was not logical to expect this to change much, because we were just shooting a minimal amount of material through the gates, and no matter how big they were we would not see the impact without putting more volume through the gate. We then shot the whole part, and to our amazement the overall pressure drop was now down to 17,000 psi, despite the pressure-drop study still showing the same 9000-psi pressure loss through the gate. Enlarging the gates increased the volume of material that could move through the gate in the same amount of time, allowing the cavity to fill more easily.

Most would assume that the first enlargement of the gates from 0.020 × 0.080 in to 0.030 × 0.080 in was a 50% increase in gate area, and would allow a comparable increase in flow volume per unit time. But you could also argue this was an even greater increase in effective flow orifice. I have no idea of how thick the skinning of plastic around the gate would be, but let's suggest 0.005 in per side. This may be a stretch, but I just want to paint a picture. If we had skinning of 0.005 in per side at the original 0.020-in gate thickness, the remaining flow-channel thickness would be 0.010 in. With the new 0.030-in gate size, the flow channel would be 0.020 in, a 100% increase in effective area and potential flow volume. With this theory, going to 0.040 × 0.080 in gate size would leave a flow-channel thickness of 0.030 in, an increase of 200% over the original size.

You really need to keep an open mind about when you need larger gates and when you need smaller gates. As mentioned earlier, typically reducing gate sizes can eliminate other defects and molded-in stress and in some cases reduce gate seal time.

Many people tend to focus on runner size as much as gate size. Although we agree that the runner should always be a consideration, from our experience 99% of the time the problem is not the runner but the gate or hot-drop orifice. It depends on the length of the runner; if the runner is short, we really do not care if it is round or square, because it has minimal impact on the pressure loss unless it is extremely undersized. Runners in most cases are much larger than they need to be, contributing to excessive waste and in some cases additional cycle time with increasing gate seal times.

Case Study: Hot Drop Restriction

On a part running glass-filled nylon the process was pressure limited at 24,000 psi. Based on the pressure-drop study showing relatively little pressure loss through the runners and gates, it was assumed not much could be done to remedy the situation. This tool had four cavities and a hot runner with two hot drops, each feeding a cold runner and one gate to each cavity. The parts in this case had a long flow length, which also pointed to the ▸

pressure loss being in the cavity as the study suggested. The process was pressure limited when the part was 50–60% full and was not able to bury all the glass on the part at the end of fill. The gates were 0.080 × 0.125 in and we opened them up to 0.187 × 0.080 in. We saw a very slight improvement but the part was still pressure limited at 80% full.

Our focus then went to the hot-drop orifices, which were at 0.080 in or 2 mm. We opened them up to 0.14 in (3.5 mm) and our pressure drop subsided to 16,500 psi. Again, this approach increased the volume of plastic to the part and ignored what the pressure-drop study was suggesting. This was eye-opening and cast new light on what education in the plastic industry has taught for years, and the lack of attention to the influence of gates or hot-drop orifices. In some cases, it just comes down to reducing the restriction and increasing the volume of material flow, and not always thinking about the psi pressure drop.

4.3 Mental Picture Volume versus Pressure

Let's use a hydraulic cylinder size as an example. A 1.0-in diameter bore in that cylinder has 0.785 in^2 of surface area. A cylinder with a 2-in diameter bore has 3.14 in^2 of surface area in the bore, which is 300% more area. If the hydraulic pressure being used was 1,000 psi, the overall force would also be 300% greater, at 785 lb of force with the 1-in bore vs. 3140 lbf with the 2-in bore. But even though the amount of force is 4 times greater, along with 4 times the volume, the pressure per square inch is still the same at 1,000 psi. If the cylinder bore were instead the gate size or area, the increase in potential volume flow and force into the part would also be 400% and the measurement of injection pressure would show no increase when in actuality the volume flow and process have drastically changed.

Let's use some realistic gate sizes and volume changes. Going from a 0.020 × 0.080 in gate to a 0.040 × 0.080 in gate has at minimum a 100% increase in potential volume flow and force. And going from a 0.050-in diameter to a 0.100-in diameter gate has an increase of at least 250% in potential volume and force.

Again, this consideration of gate area and volume is important mainly when there is an issue with a pressure-limited ability to fill or pack the part adequately. Our current methods using pressure-drop studies focus on one parameter, and assumptions of what is relevant can blind us to the reality of what is happening inside the mold.

5 Mold Hydraulics

5.1 Hydraulic Uses

Hydraulics are used for a number of reasons in the injection molding industry. They are used for slides, core pulls, ejection, reverse ejection, gas assist over flows, gas assist valve pins, valve gates, locks, unscrewing molds, etc. [1].

5.2 Set and Pull Pressure

In situations where hydraulic cylinders are used set/pull pressures and speed must be considered. Set/pull pressures and speed can directly affect downtime with leaks and repairs. Pressure and speed can be your friend and foe. Time is money so speed is your friend from this viewpoint. But whenever speed is causing an impact with a mechanical feature that you can hear when it is set or pulled, it is causing stress, wear, and at some point broken components with T-slot couplers or threaded rod ends. There are a few options to protect components from the negative impacts of speed. Cushions can be added in the cylinder. It is preferred to have your cylinder bottom out versus your component on the pull direction. On set position your component needs to be home so you typically do not want your cylinder bottoming out. Evaluate how speed will impact your component and potential failures.

Hydraulic pressures can also have a negative impact on components. On the set position, if the component is relying on hydraulic pressure to overcome plastic/cavity pressure a high set pressure may be required. If the component is locked in place with a lock angle on the stationary half it does not need to use excessive hydraulic pressures, but rather just enough pressure to move the component. Also with the pull action only enough hydraulic pressure is required to pull the component back.

There have been many failures over the years with the root cause of using excessive hydraulic pressures. In most cases this occurs with the pull position with the attachment point of the cylinder rod and component getting stretched/stressed whereas in the set position they are compressed. One other thing to consider when using T-slot couplers is that it is important not to have sharp corners on the T or the slot, because sharp corners will increase the chance of failure from stress.

5.3 Cylinder Sizing

When the cylinder is being used to resist plastic pressure, it is critical to understand proper sizing. If the component is using a locking angle with the stationary half you just need to size the cylinder to move the component. When the hydraulics need to withstand plastic pressure the math is very simple and no textbook is required to determine cylinder size.

To correctly size a hydraulic cylinder on a mold follow these steps:

First the mold designer really needs to understand what are the potential maximum plastic pressures so he/she can accurately calculate the plastic pressure the cavity surface will have on the component. Many will look at just pack/hold pressure but you need to look at the peak plastic pressure. Also, you cannot just base it off the material being used; as mentioned earlier there are a few things that can drastically impact the pressure. If you are unsure, round it up; never round it down if you want a robust tool and process. Once you have determined the maximum plastic pressure the rest is easy.

The next step is to accurately calculate the cavity surface that is on the surface area of the component. Once you have determined the cavity surface area you multiply that by the maximum plastic pressure we discussed. This result will be the pressure that you will need to design your cylinder around.

The next step is to determine what bore size your cylinder will be. I typically will design the bore size so it exceeds the cavity pressure force by 1.5 times to make sure it is robust. For example, if our expected cavity pressure is 10,000 psi and the cavity surface area of the component is 1 in^2 we know that there would be 10,000 pounds of force acting on the component. If we multiply the 10,000 by 1.5 ratio we know we need approximately 15,000 pounds of force to counteract the cavity pressure.

To determine the hydraulic cylinder bore size needed we need the surface area of the cylinder bore size and the hydraulic pressure being used on the machine or hydraulic auxiliary pump cart. In this example, let's say we will have 2,000 psi of

hydraulic pressure. Then we calculate the surface area of the bore with $3.14 \times R^2$. If we use a 3-inch diameter bore cylinder we take the radius, and calculate $1.5 \times 1.5 \times 3.14$, which equals 7.065 in^2 of surface area. We then multiply this by the 2,000-hydraulic psi being used. This equates to 14,130 pounds of holding force. So in this scenario a 3-inch bore cylinder should provide a robust condition.

Just as for the cavity pressure, it is very important to know exactly what hydraulic pressure is going to be used. This is often overlooked or assumed. Keep in mind that on some machines the core pressure can be set up as pressure limited with a pressure relief valve. So even though you are changing the pressure on the controller the actual pressure is not going up if the relief valve is set lower. So what your controller says may not be actually correct. The only way to accurately tell is to put in an inline pressure gage to measure the pressure. In the case above, if the pressure is limited to 1,000 psi you would blow back the cylinder because we would only be applying 7,065 pounds of force to counteract the 10,000 pounds the cavity is applying to the core surface. I have witnessed this many times and therefore had to upsize cylinders. In one case upsizing the cylinder was not an option; instead we had to add a second set of cylinders to activate a lock to hold the component in place, which made the tool more complex with a suicide condition.

One other thing that needs to be considered is the psi rating of the cylinder itself. If the cylinder is only rated for 1,500-hydraulic psi and you are using 2,000-hydraulic psi you will run into cylinder failures with leaks and blown seals.

In a case where your cylinder is not rated for the hydraulic pressure being used, for example with a cylinder rated for 1,500 psi and using 2,000-hydraulic psi, you can add an inline pressure limiting valve on the mold to protect the cylinder from failures. Also in the case where the cylinder may be rated for higher pressure that is not required you can add this pressure limiting valve to protect the moving component from failure. Also with some hydraulic valve gates the cylinders are not rated for high pressures. One other aspect to consider for bore size is that if you are using a double-ended rod the diameter of the rod will need to be subtracted from the bore size because it reduces the surface area on the set/forward position.

There are also cases where a self-locking cylinder can be used if you do not have the real estate for the bore size needed to counteract the cavity pressure. These types of cylinders have a physical mechanical lock on set that locks the hydraulic piston in place to prevent blow back. I have many tools with these cylinders and they are reliable with minimal maintenance if used properly. They do have set/pull pressure specifications that you need to be aware of when using them. Timing with your component in the set position is critical for the mechanical lock feature but a very simple process when installing on the mold.

5.4 Hydraulic Cores or Slides

When using hydraulics for core pulls that have shut-offs on the cavity, the shut-off should not be the positive stop that takes all the impact of the hydraulic pressure in the set position. It is best to have a step or ledge on your component that bottoms out at the same time as the shut-off to protect it. Furthermore, at times these shut-offs need to be vented and without the added protection they will just hobb shut over time.

Hydraulics are also used with slides that require excessive travel when the standard mechanical horn pin cannot provide the amount of travel needed. In many cases the slides will have a lock angle with the stationary half locking it in place against the plastic pressure where the hydraulic pressure is not needed to withhold cavity pressure.

With hydraulic driven slides extra caution is needed when there are low draft shut-offs between the slide and the stationary half. This can contribute to galling on the shut-off with the slide being locked forward with hydraulic pressure. In these situations when adequate draft cannot be added there is a simple device you can add to protect your component on the set position if there is a lock angle on the stationary half of the mold that locks your component forward. If you use a T-slot coupler you can create extra travel in the machined slot and add a spring to hold it in the back position. Normally this will be set up where the component is 0.030–0.060 in from the home position when the cylinder is bottomed out. When the mold closes the lock angle on the stationary half will push it in the home position while collapsing the spring. This will protect the shut-offs from galling. This method can also be used when the component has critical shut-offs of the moving half to protect them with set speeds and pressures.

5.5 Multiple Sequences

When hydraulics are used on a tool there is a good chance that the tool will have a suicide/crash condition that can be costly to repair if this occurs. Also when using multiple core/hydraulic sequences this situation is typically a risk. When using multiple hydraulic sequences, it is important to label them, color code them, or better yet use different size fittings for each sequence so they cannot be set up improperly in the press. Using male and female connections for set and pull can help distinguish them at setup. It is also important to have a core sequence plaque on the mold to show the sequence of the cores to assist with machine setup [2].

5.6 Mounting

When using mounting plates to attach the hydraulic cylinders to the mold make sure the plate and bolts are robust enough to prevent flexing and broken bolts from the stresses. There have been many failures over the years due to inadequate mounting plate thickness and bolt sizes. Whenever the cylinder rod is attached to the component with threads make sure the hole is tapped perfectly straight/square. Also consider doweling the plates in position so everything stays on center of the cylinder. Threads can work loose over time so lock tight, or a set screw can be used to reduce the chances.

Whenever possible try to protect a cylinder and the switches on a mold. There are many examples of damage to molds due to a lack of protection for mold components.

5.7 Switches

There are two styles of switches that are typically used with hydraulics for set and pull position, mechanical switches, and proximity switches. Both types have advantages and disadvantages and every molder must make an appropriate choice.

Mechanical switches are easy to adjust and verify if they are actuating. They are normally mounted outside of the mold components and provide for ease of access. Mechanical switches are not as compact and clean as proximity switches.

Proximity switches are frequently used because they are compact and part of the cylinder. These typically read magnetically and sense the position of the piston, either directly on the piston itself with the sensor bolted in a hole in the wall of the cylinder or through the cylinder wall when made of aluminum or stainless steel. When using sensors that are bolted in a fixed position, timing of the component for the set and pull position is critical so the piston is in the correct position and the sensors can read it in the set and pull position. Sensors that read through the wall of the cylinder can be adjusted to any position so timing of the component is not as critical. However, when using aluminum cylinders you need to make sure they will be adequate with the cavity and hydraulic pressures. Try to stay away from these in high-temperature or high-cavity-pressure situations because they are not as robust. Also when using proximity sensors you need to take into consideration the temperatures they will be experiencing. Most will not read properly when going over 150 °F.

References

[1] This chapter first appeared in Plastics Technology Magazine, “Tooling: The Impact of Hydraulics on Tool Design”, Kerkstra, Randy 7/16/15

[2] Originally appeared in Plastic Technology Magazine, “Tooling: The Impact of Hydraulics on Mold Design”, Kerkstra, Randy 8/17/15

6 Mold Texture and Polish

6.1 Mold Texture

There are a wide variety of texture patterns available for molds to provide many unique looks. The basic idea behind a texture is to change the appearance of the part beyond just a plain flat plastic surface. It is important to understand the impact of texture on the finished part because texture may improve part appearance but may also cause additional problems such as sticking and drag marks.

The steel surface finish and texture will affect gloss, cosmetics on the part, and part sticking issues. It is also plastic-material specific regarding cosmetics or sticking issues; what works for one material may not work for another. The common reaction to a part sticking issue is to shine up the cavity surface, although on smooth surfaces with some materials this will have a negative impact.

So what exactly on a microscopic level affects gloss on the plastic part? It comes down to the geometry and depth of the texture and the surface finish of the texture/steel itself via polishing or blast media. To understand exactly how this works we will need to go under the microscope where the naked eye cannot see.

We mentioned two things that affect the gloss: the geometry, and the surface finish. And remember we are molding the plastic to the steel so the color of the tool steel has absolutely nothing to do with gloss. The steel can appear to have a different color when polished or roughed up but it is the surface finish and how it reflects light that changes the color appearance; for example, Figure 6.1 shows various gloss levels on the same piece of steel. The color of the steel appears to vary but the color is actually the same; it is how the various surface finishes reflect light differently that affects this [1].

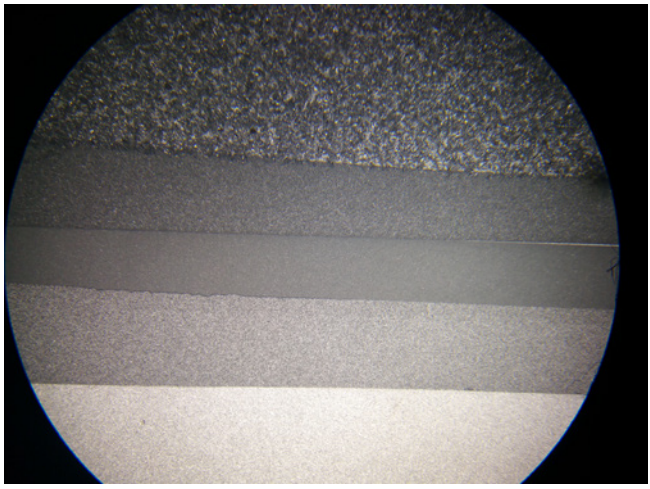


Figure 6.1 Steel with various blast media

Just think of a lake on a very quiet calm day and how the sun reflects itself on the calm water. It can produce a glare that burns the eye. Now on another sunny day but with light breeze causing soft ripples you can still get a reflection but the ripples distort it. Now let's move to a sunny day with a brisk wind causing waves; the sunlight is even more distorted. Two things are happening. First think of the water as the texture; the flatter and smoother it is the glossier the part, and conversely the deeper the texture (waves) the duller the part. The other thing to note is when the texture is deeper you are creating more surface area with the peaks and valleys in the same amount of the 2D profile area of the part.

Textures have come a very long way the past decade or so and have some very complex geometries. The depth, patterns, crispness, or peaks of the valleys of the texture geometry have a big impact on part gloss. There is also a process called microetching (see Figure 6.2), which breaks up the geometry of the texture to make a duller part. This is just like putting waves on the smooth water surface to reflect/distort the light.

Now that we understand how the texture geometry can affect gloss let's go a step deeper. Let's consider the geometry of the surface finish on the texture itself. We are now getting into the micro level but the surface finish of the texture geometry is just another level of texture with flat and smooth versus peaks and valleys. This is where blast media or polishing come into the picture, in particular in how they affect gloss. The main two blast media used in the industry to adjust gloss are aluminum oxide and glass bead. Silica sand and other media are used but no matter what blast medium you use it is the effect they have on the surface finish that influences the gloss. Some also refer to this process as vapor honing but it is all the same.

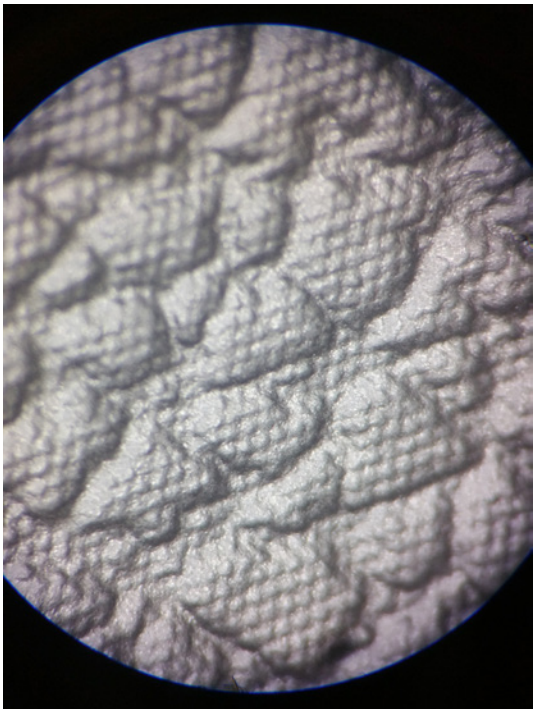


Figure 6.2 Magnified micro texture

When acid etching of a cavity is complete, to create the texture a blast medium is used to clean up the surface and achieve the desired gloss for the specified gloss of the plastic part. Aluminum oxide is an abrasive, basically a rock with sharp edges creating peaks and valleys on the surface finish of the texture geometry, which will produce a duller finish on the plastic part. There are numerous sizes of aluminum oxide, referred to as grit size, that impact the depth and surface finish also. Glass bead is round and smooth like a marble, and will produce a smoother finish on the texture and a glossier part. So the aluminum oxide surface finish on the texture will provide the dullest gloss on the plastic part and the glass bead surface finish on the texture will provide the highest gloss with media blasting.

The two media can be combined at different ratios to achieve any gloss between the low and high gloss they produce alone. To achieve the mid-range between the two media you will need to have a 1:2 ratio: one part aluminum oxide and two parts glass bead. The aluminum oxide has a bigger impact on the surface finish than the glass bead, which is why the ratio is not 1:1. Various blends with ratios that go from 1:1 to 12:1 are not uncommon. It all depends on what type of gloss you are trying to achieve. Typically try to use three media variants: lowest (aluminum oxide), mid (1:2 ratio), and high (glass bead) gloss. When the gloss is critical and cannot be achieved with these three go with what we refer to as custom blends/ratios in between the three.

When running glass-filled materials, gloss will change over time from erosion of the surface finish of the texture or eroding the texture itself. So in this case consider using hardened tool steel, coatings, or surface treatments.

For visual defects or cosmetic issues typically a higher gloss level on textured parts will produce a better looking part. When gloss levels get low (for instance under 3 on a 60° gloss reading) the dullness of the molded part will tend to magnify defects.

6.2 Mold Polish

Molds can be polished to a wide variety of levels depending on the requirements of the finished part. Polishing is conducted in a series of increasingly fine grit levels to achieve the desired results up to an A1 mirror finish (see polish chart in Table 6.1).

Table 6.1 Polish Chart

Finish	Polishing medium
A1	#3 diamond
A2	#6 diamond
A3	#15 diamond
B1	600 grit paper
B2	400 grit paper
B3	320 grit paper
C1	600 stone
C2	400 stone
C3	320 stone
D1	#11 glass bead
D2	240 aluminum oxide blast
D3	#24 aluminum oxide blast

It is vital to keep in mind that not all materials will interact well with a polished surface. For instance, materials such as thermoplastic polyurethane and thermoplastic elastomers will often stick hard to polished surfaces and even polypropylene will show a tendency to stick when exposed to highly polished surfaces. When dealing with sticky materials like those mentioned, often times a blasted surface will be required to help break the vacuum of the part sticking to the mold. In cases where a glossy finish is desired a 320 paper finish will often work well as a compromise that provides adequate release.

Note that as the polish of a mold becomes finer the cost increases significantly as does the effort required to maintain the high polish. When dealing with diamond polished surfaces contact with the mold surface and any foreign objects including mold wipes should be minimized to avoid scratching the surface. If there is any contamination on the mold wipe or the mold surface the chances of damaging the diamond polished surface are very high. For lens molding often times the way to clean a mold is spraying with a can of mold cleaner and blowing the mold cleaner from the surface with an airline.

Maintenance of a diamond polished surface should be conducted by trained personnel who can determine the proper polishing media and techniques. Diamond polishing may require wood or felt for application and the wrong choice of paste or technique may cause substantial damage to the mold.

A draw polish is a polish method where the final step of polish is in the line of draw. Using a draw polish will normally provide better release of a molded part. There are many cases where a rougher draw polish provides a much better release than a finer polish, so do not assume that polishing to a finer level will eliminate sticking. Remember that even an A1 diamond polish has microscopic scratches from the act of polishing, and these scratches may inhibit release of the part.

References

[1] First appeared in Plastics Technology Magazine 3/31/15 “Tooling: Clearing Up the Mysteries Of Mold Texture”, Kerkstra, Randy

7 Venting

Proper vent design based on the material being molded and the process is critical. Venting of a mold allows gas to escape from the cavity as the plastic fills the mold. It is important to remember that the cavity is filled with air at the start of fill and the plastic will usually have outgassing during the molding process, so if the mold is not vented properly the gas will become trapped in the mold. Trapped gas will lead to defects including burns, short shots, splay, and poor gloss replication.

In addition to basic venting to remove trapped gas from a mold cavity it is vital to vent any weld line locations (see Figure 7.1).

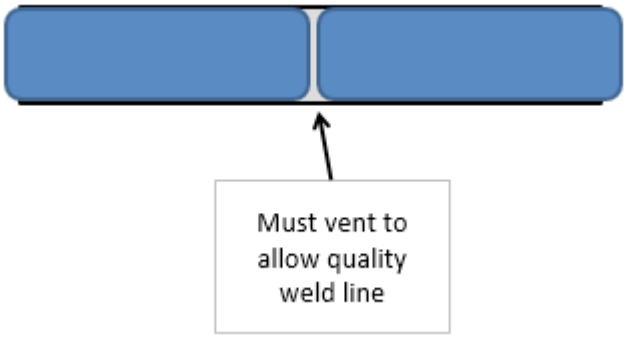


Figure 7.1 Weld line locations are critical for venting

■ 7.1 Venting Basics

Every vent has the three dimensions depth, width, and land. Figure 7.2 shows a cross section of the vent depth and land. Also shown in Figure 7.2 is the relief channel of a vent. Vents should always be considered in area rather than simply depth. The width and depth provide the total area for the gas to evacuate.

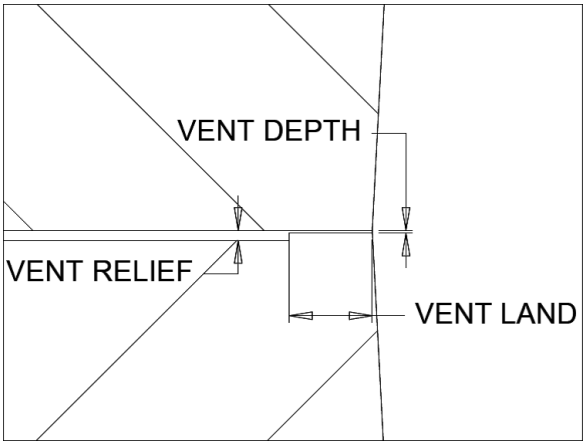


Figure 7.2 Vent diagram

Vent depth is dictated by the material being molded as well as the part design and the process. The viscosity of a given material will impact how deep a vent can be. In the case of a material such as polycarbonate vents can be 0.003″ to 0.004″ deep, but in the case of a material like PPS 0.0005″ would be a more appropriate vent depth. Always consult the recommended vent depth given by a material supplier for the material being molded. Vents that are too deep will result in flash but too shallow ones will not allow adequate venting. Experience has proven that normal vent recommendations are accurate for about 80% of molds but based on part design and process you may need to deviate from the standard.

To determine if deeper vents will improve an issue one of two things can be tried. The first is to reduce the clamp tonnage to allow better venting, and even taking this to the point of slight flash will help determine the effect of the improved venting. Another way to check if deeper vents will help is to add tape to the bearing surface of the parting line near the defect location, as the tape will simulate an increased vent depth. Be careful when adding the tape to ensure that the tape stays away from the edge of the cavity, because this will reduce the chances of damaging the cavity parting line.

Vent land length is the distance from the cavity to the vent relief channel. In most cases vent land length should not exceed 0.080″ and can be reduced to as little as 0.030″ in problem areas. If venting land length is excessive the vents will have a tendency to experience excessive buildup of out-gas by-products.

The relief channel of a vent is installed outside the vent land length and provides a dump for the out-gassing to exit the mold base. Relief channels should be a minimum of 0.010″ and can be as deep as 0.030″ depending on the gassiness of the molded material. Vent relief channels must be directed to the exterior of the mold to ensure the gas fully exits the mold base. In cases where the vents are on the

interior of a framed part the relief cannot be directed to the exterior of the mold base, and in this case the relief should be directed to a vent hole that allows the gas to exit through the back of the core retainer plate. Always add a pin to each vent hole to ensure that the hole does not flash over.

Vents should normally be as wide as possible. Traditional vents have been cut in at a given cutter width as shown in Figure 7.3. Venting is not maximized or even optimized with this type of vent, and many times an end of fill vent will be cut in a mold only to find that the true end of fill is slightly offset from the vent location. To better optimize venting and provide optimized venting, cut vents as shown in Figure 7.4. This improved venting method provides much more venting area for gas escape and will allow for better process windows and less buildup in vents. Another option to consider is full perimeter venting to ensure the best possible evacuation of gas. Another advantage to the venting method shown in Figure 7.4 is that it allows the mold to maintain more bearing surface.

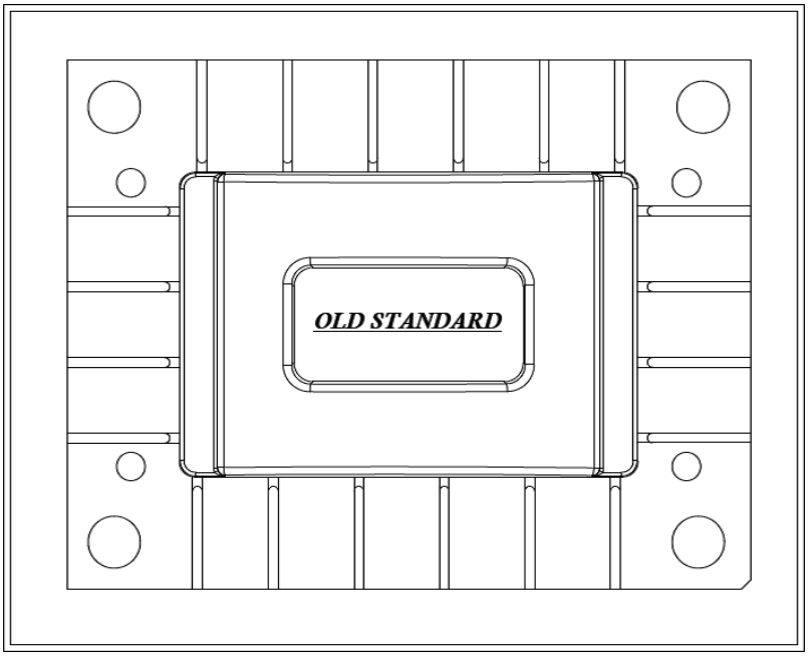


Figure 7.3 Old venting standard

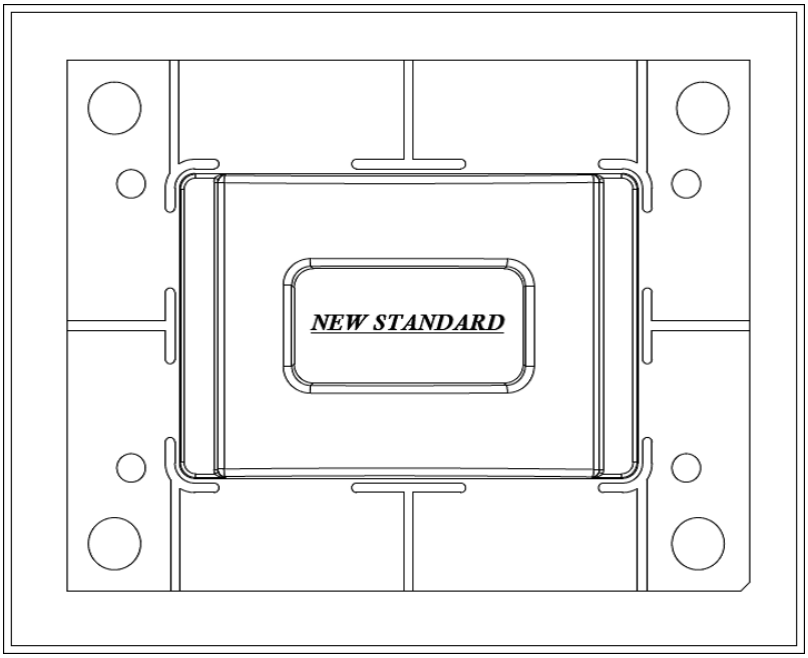


Figure 7.4 New venting standard

The parting line bearing surface also has an impact on how long the vents stay open. Excessive parting line relief will result in coining of the parting line shut-offs and closed vents. Excessive vent relief channels can also take away area from the parting line bearing surface. Smaller vent channels that run along the edge of the cavity with fewer exiting channels to atmosphere provide more venting and reduce the amount of parting line relief providing a more robust design for venting issues. Figure 7.5 shows a mold with traditional vent channels that resulted in a reduction in bearing surface and repeated issues with vents closing off due to steel coining. Also make sure that excess clamp tonnage is not used on a mold, or the vents will be coined in due to compression of the bearing surface steel.

When venting ejector pins there are different techniques; see Figure 7.6. A ring vent provides much more venting compared to the old standard with vent flats; see Figure 7.7. With ejector pins vent land should start at a maximum of 0.060″ and with stationary pins the vent land should start with a maximum of 0.040″ and as little as 0.020″. Ejector pins are somewhat self-cleaning and can be accessed for cleaning when ejected forward. Stationary pins are not accessible without removing so it is critical that the vent land is not excessive resulting in quicker plugging.

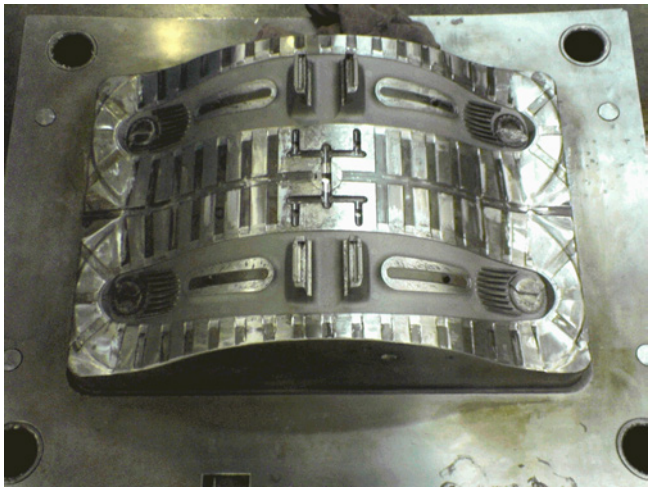


Figure 7.5 Traditional venting with lack of bearing surface

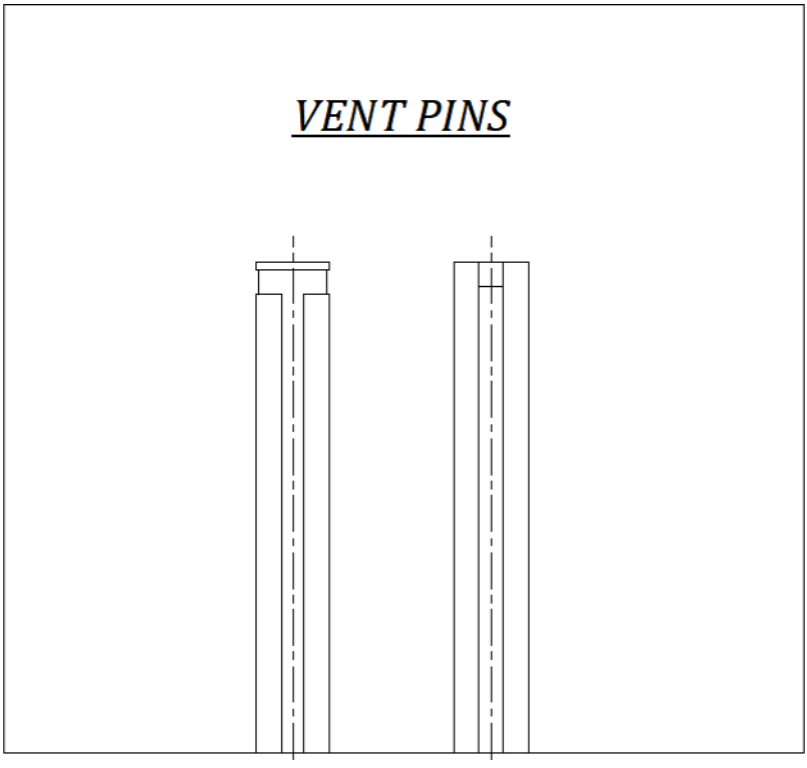


Figure 7.6 Pin venting detail

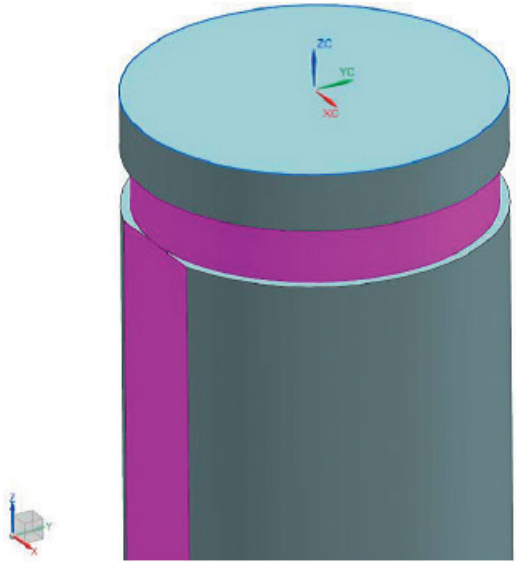


Figure 7.7 Pin vent with ring relief

Porous steel or sintered vents are also an option with difficult part details and backfill areas. It is important to follow proper instructions with machining and cleaning, and to have the proper relief channels to reduce plugging of the pores. Porous metal inserts must be kept clean to ensure proper venting and inserts may need to be removed for cleaning or replacement on a regular schedule. One other consideration with backfill areas is using a stationary pin and sleeve with ring vents if porous steel is not preferred.

Vent inserts should be designed whenever possible for removal and cleaning in the molding machine. Vent land on inserts should be a maximum of 0.040″ and vent depth at maximum depths based on the material and process to reduce buildup. Also bear in mind that venting can and should be added to lifters and slides whenever possible.

■ 7.2 Alternate Venting

If additional venting assistance is required a vacuum venting system can be added to a mold to allow the gas to be drawn out of the mold. This can allow for the best overall venting possible in a mold. In most cases it is recommended to seal the mold with gaskets and route all vent channels to the vacuum line. In some cases adding a vacuum to a mold even without gaskets will lead to improved venting.

Another way of maximized venting is to actually create an overflow where the plastic will come together at a weld line and flow through a thick vent into an overflow tab. This method can be very effective at improving weld lines on glass-filled materials. Figure 7.8 shows an example of overflow venting. Think of this overflow as a reverse gate, and in fact using a sub gate design at the end of fill weld line can create a nice overflow that automatically separates during ejection.

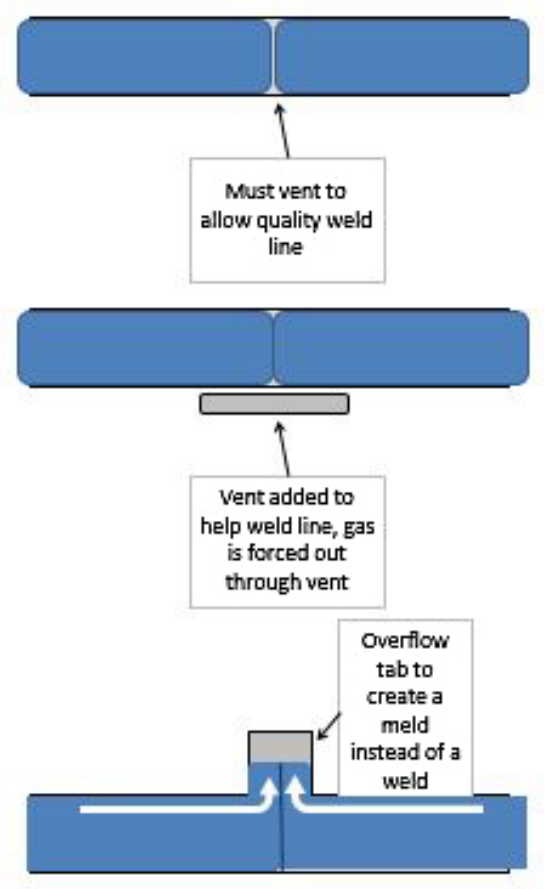

Must vent to allow quality weld line
Vent added to help weld line, gas is forced out through vent
Overflow tab to create a meld instead of a weld

Figure 7.8 Overflow venting

Actuated vent pins can be used where an output signal from the molding machine opens and closes a vent within the mold. These signals can be configured in numerous ways such as using configurable I/O or even valve gate outputs. Cavity pressure instrumentation could be used to determine when to trigger the vent to close to prevent flash.

Another option for increased venting is to use a molding machine with injection compression or coining option. In injection compression molding the mold is not completely clamped during the initial filling of the mold, which allows for easy evacuation of the gas within the cavity. The mold locks up under tonnage after filling to a point and provides for the ability to pack out the part using the force generated by the clamp (the compression side of injection compression molding).

7.3 Venting Conclusion

Venting is critical for producing quality parts. Spend the time during mold design reviews to ensure that adequate venting is provided on all molds. Vents must be maintained during the life of the mold to provide proper gas evacuation and should be kept clean during all mold runs. Checking for vent cleanliness is one of the 4M Basic 8 troubleshooting steps and is a quick step that often eliminates many processing problems. Do not process around venting issues: the problem will only get worse over time!

8 Machine Performance

The methods discussed in this chapter all reference to an ANTEC paper written by John Bozzelli, Rodney Groleau, and Norm Ward "The Machine Audit: A Systematic Evaluation of Injection Molding Machines: How To Tell a Good Machine—Old or New" [1].

The simplest thing to keep in mind is that a machine must do what we want when we want it. If a machine is set to produce 700 psi hydraulic hold pressure it should provide 700 psi every shot day after day. Many times, it will be said that a given mold does not run well in a given press but is fine in another identical press. When running into this type of problem the root cause of the machine difference must be determined. If the machines are not producing the same results then something is wrong. If the performance of the machine is not measured a processor is relying on hoping a press is doing what is needed. A good rule of thumb is not to process around machine issues, because this only narrows the process window and will eventually cause part problems.

Walking up to a machine and giving a look over does not tell how well a machine will perform. See Figure 8.1; is this a quality machine capable of providing repeatable processes? It is not possible to know unless a systematic approach to machine evaluation is used.

The example shown in Figure 8.2 of a machine testing report card that documents the various tests is one easy-to-use format. Using a simple green and red rating with an overall A–E grade is a simple and visual way to grade and compare all the machines in the shop.

Figure 8.1 Typical injection molding machine

Machine Testing Scorecard

Machine: #6 Date: 8/23/05

Results

Velocity linearity	41.2%	Dynamic check val	3.0%
Load sensitivity	3.3%	Fill time variation	1.9%
Hydraulic response time	0.16 sec/1000psi	Total Points	23 out of 27

Grade B

Note: Fill in all yellow highlighted boxes

Velocity setpoint	Shot size	Decomp	Transfer position	Fill time	Velocity actual	% difference
0.1	1.55	0	0.9	7.22	0.09	10.0%
0.2	1.55	0	0.9	3.77	0.17	13.8%
0.4	1.55	0	0.9	1.89	0.34	14.0%
0.8	1.55	0	1	0.91	0.68	15.1%
1.6	1.55	0	1	0.52	1.06	33.9%
2.75	1.55	0	1	0.41	1.34	51.2%
					Range	41.2%

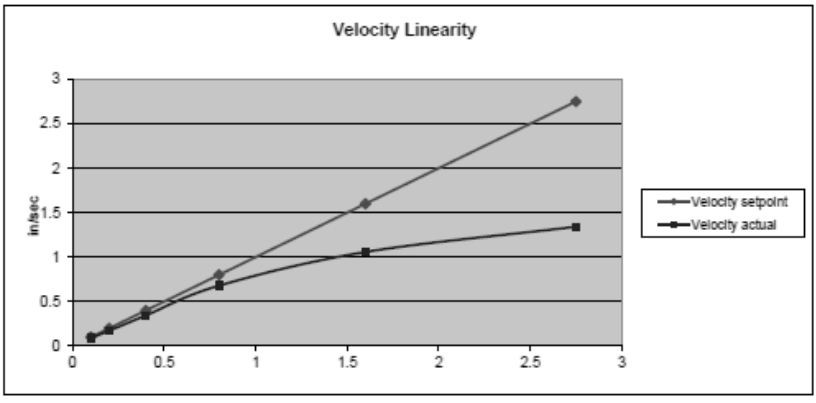


Figure 8.2 Example of a machine testing report card page

8.1 Load Sensitivity

Load sensitivity or load compensation is a test that measures the ability of a machine to react to viscosity change. In general, it is a well understood fact that all materials will have viscosity variation over time. Load sensitivity tests the ability of the molding machine to provide a consistent fill time even if viscosity varies.

To conduct a load sensitivity test follow these steps:

1. Run a standard fill only shot without second-stage pressure on. This fill only shot should be 95–98% full at transfer.
2. Record the pressure at transfer and fill time. These will be FTm and FPm.
3. Back the barrel away from the mold. The next shot needs to be an air shot run at process with the barrel backed off. For some machines the barrel forward switch must be moved to accommodate this new position; some machines allow turning off the carriage whereas others require the carriage to be forward. On machines where the barrel must be forward against the mold use an RJG purge plate to allow an air shot to be run in semi-automatic operation.
4. Cycle the press allowing the machine to shoot into the air.
5. Record the pressure at transfer and fill time for the air shot. These will be FTa and FPa.

Then the load sensitivity is calculated according to Equation 8.1.

$$\left(\frac{\left(\frac{FTm - FTa}{FTm}\right)}{\left(\frac{FPm - FPa}{1000}\right)}\right) \times 100 = \%\ load\ senstivity \tag{8.1}$$

Note: use 1,000 for the denominator when using hydraulic pressure; use 10,000 for the denominator when using plastic pressure.

The accepted standard is that this number should be close to zero. If the value is greater than 5%/10,000 psi the machine will not be capable of maintaining consistent fill velocities as conditions vary. Machines that are load sensitive will not provide consistent fill times, which means that the process will not be repeatable over time. If fill times are not consistent all other parts of the process will also be affected. See Figure 8.3 for an example of a load sensitivity form.

Time/pressure	Value
Fill time mold	3.69
Fill time air	3.77
Fill pressure mold	953
Fill pressure air	294
Load sensitivity	-3.3%

Figure 8.3 Example of load sensitivity test

A machine may perform better at certain velocity ranges than others so it is often recommended to conduct load sensitivity tests at different fill speeds [2].

8.2 Dynamic Check Valve Study

Check valves or non-return valves tend be a spot of process variation over time. A non-return valve can become worn, damaged, or have contamination trapped in the shutoff seat. If the non-return valve is not seating consistently variation in the amount of plastic injected during the first stage will be the result [3].

To conduct a dynamic check valve study do the following steps:

1. Run 95–98% fill only shots.
2. Run 10 consecutive shots. Save every shot.
3. Weigh each shot and record the fill only shot weight.
4. Record the fill time for each of the 10 shots.
5. Calculate the % variation for the fill only weight and fill time according to Equations 8.2 and 8.3. Fill only weight should be less than 3% and fill time should not vary by more than 1%.

$$\frac{maximum\ shot\ weight - minimum\ shot}{maximum\ shot\ weight} = \%\ variation\ fill\ weight \tag{8.2}$$

$$\frac{maximum\ fill\ time - minimum\ fill\ time}{maximum\ fill\ time} = \%\ fill\ time\ variation \tag{8.3}$$

Also look for variation in cushion over time especially in any cases where the cushion reaches zero. Variation in cushion often indicates an issue with a non-return valve. Other indicators of non-return valve issues are screw rotation during inject forward and large amounts of travel after transfer.

To determine if the problems stem from the non-return valve or a worn spot in the barrel try moving the shot size and transfer position back approximately 25 mm and doing the 10-shot study again. If the fill only weight variation is better after changing the shot size and transfer position there is probably a worn spot in the barrel that is causing the problem rather than the non-return valve.

If a non-return valve appears to be an issue the end cap of the press should be removed and the screw tip and non-return valve should be inspected for wear, damage, or contamination. If an issue is detected the non-return valve should be replaced.

8.3 Velocity Linearity

Processors often assume that the setpoint that they enter in a machine controller is what the machine will provide. A classic area where this may not be the case is velocity. To determine if the machine is providing the velocity that is requested follow these steps:

1. Set the process to run a fill only shot, no second-stage pressure.
2. Adjust transfer so that the part is approximately 80% full.
3. Record shot size, decompression, and transfer position.
4. Set the fill velocity to the minimal practical value.
5. Record fill time.
6. Double the velocity setpoint.
7. Record fill time.
8. Repeat steps 5–6 until machine maximum is obtained.
9. Calculate the actual fill velocity for every step (Equation 8.4).
10. Calculate percent difference for each velocity (Equation 8.5).
11. Calculate the total range of fill difference by subtracting the minimum difference from the maximum difference. Maximum range should be below 10%.

Calculating actual fill velocity:

$$\frac{\left(shot\ size + decompression\right) - transfer\ position}{fill\ time} = actual\ velocity \tag{8.4}$$

$$\frac{set\ velocity - calculated\ velocity}{set\ velocity} = \%\ difference \tag{8.5}$$

If a press does not have good velocity linearity, changes in the fill velocity will not yield the expected results. In some cases, machines will not be able to achieve the maximum value that the controller allows. The velocity linearity curve will demonstrate where the deficiencies in velocity control occur. There will also be cases where a machine may be tuned to perform well at one end of the velocity range but not as well at the other end. Figure 8.4 shows an example of a spreadsheet for evaluating velocity linearity.

Velocity setpoint	Shot size	Decomp	Transfer position	Fill time	Velocity actual	% difference
0.25	3.3	0	1.2	9.86	0.21	14.8%
0.5	3.3	0	1.2	4.7	0.45	10.6%
1	3.3	0	1.2	2.95	0.71	28.8%
1.5	3.3	0	1.2	2.36	0.89	40.7%
2	3.3	0	1.2	2.03	1.03	48.3%
3	3.3	0	1.2	1.64	1.28	57.3%
					Range	46.7%

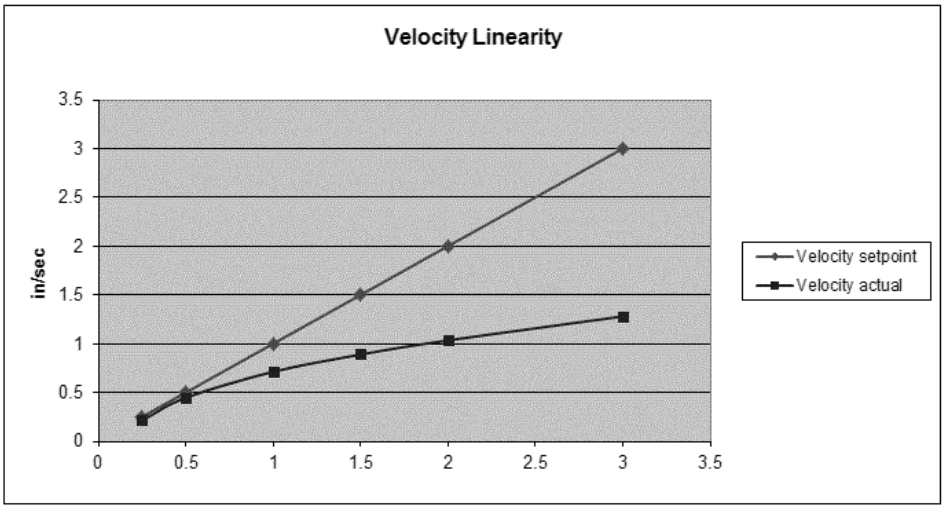


Figure 8.4 Velocity linearity form and graph; note the poor actual velocity response

There will be cases where a process technician is making velocity increases on a press but the fill time is not changing. This is a good indicator that the press is not capable of achieving this velocity (always beware of pressure limiting).

Another useful bit of learning from this test is to determine the actual fill velocity on a press that uses percentage for velocity set values. If a machine only offers percent settings it is a valuable exercise to document how the percent values correspond to actual velocities or pressures. The values can be documented on a curve which can be posted on the molding machine to allow a quick visual reference of

what an actual value is for a given setpoint. See Figure 8.5 for an example of a percent versus actual velocity and volumetric flow graph.

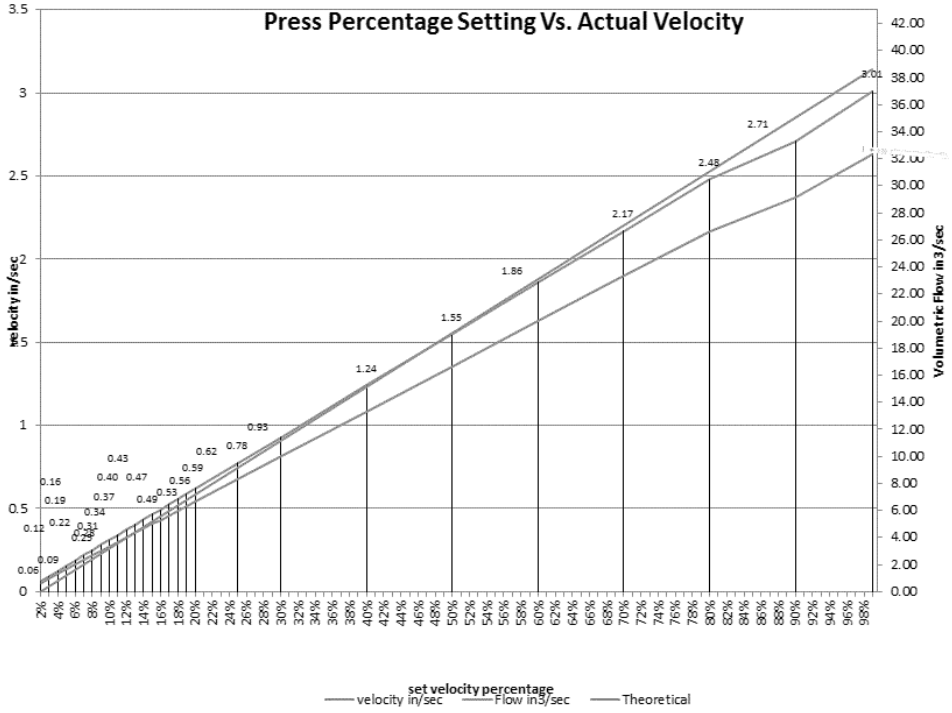


Figure 8.5 Machine % setting vs. actual in/sec and volumetric flow

Some machines will have velocity settings ranges that do not work well. The classic example is a machine that uses percent settings and a change from 18 to 19% results in a 25% change in fill time! See Figure 8.6 for an example of a machine with "dead" spots in velocity control. If the time is not taken to conduct a linearity study, this type of problem will be a mystery. Some of the process technicians on the floor may recognize that this problem exists but not have a clear vision of what is really occurring. Note that there are cases where the velocity charts provided for the machine show that there will be a jump at certain percentages; if the charts are available it may only be necessary to confirm actuals to the provided chart. This may never be an issue if the machine never runs in this particular percent velocity range; however, it is still useful to know.

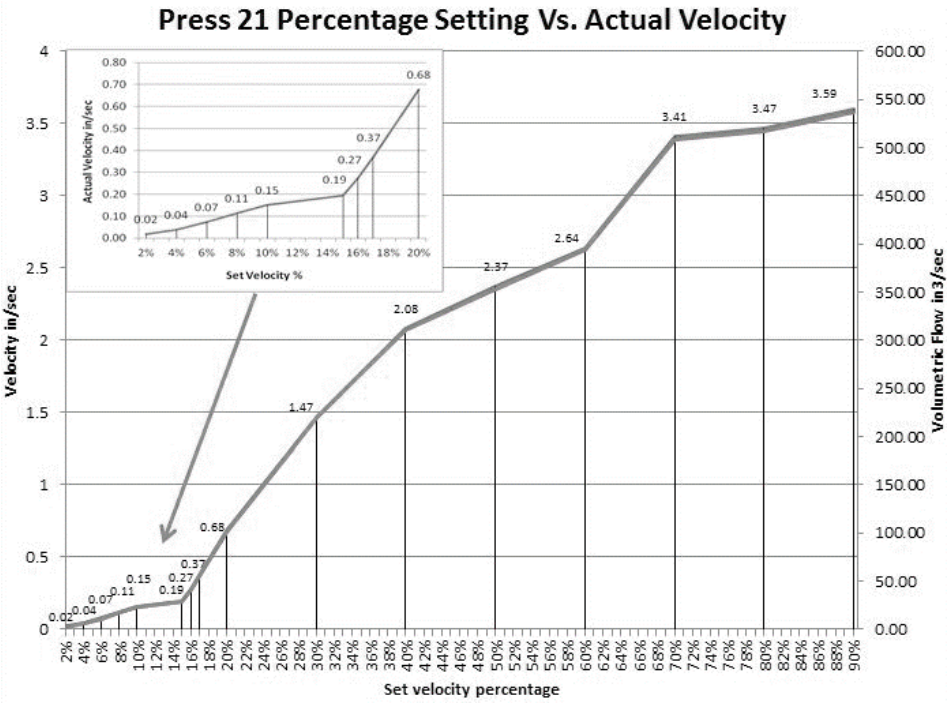


Figure 8.6 Example of non-linear velocity response

■ 8.4 10-Cycle Overlay

The basic requirement of this test is to take 10 consecutive shots and overlay the data with a process monitoring system. In an ideal case the curves from the 10 shots will overlay perfectly for all 10 shots and it will look like one single line. In the case of a machine that is not performing consistently you will be able to count individual lines across the graph (see Figure 8.7 and Figure 8.8 for the difference).

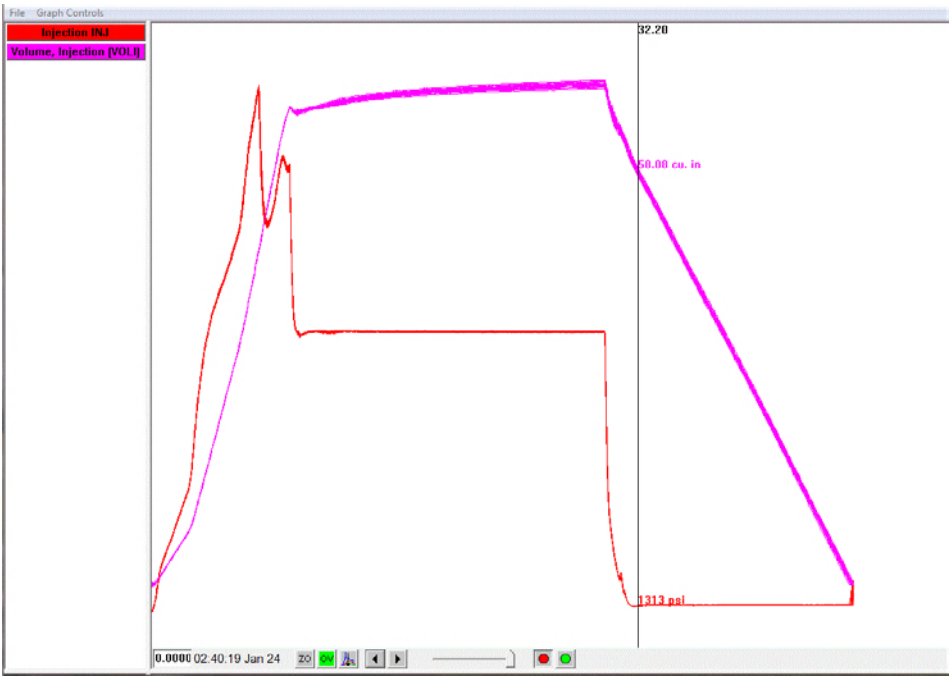


Figure 8.7 eDART® curves showing good 10-shot repeatability

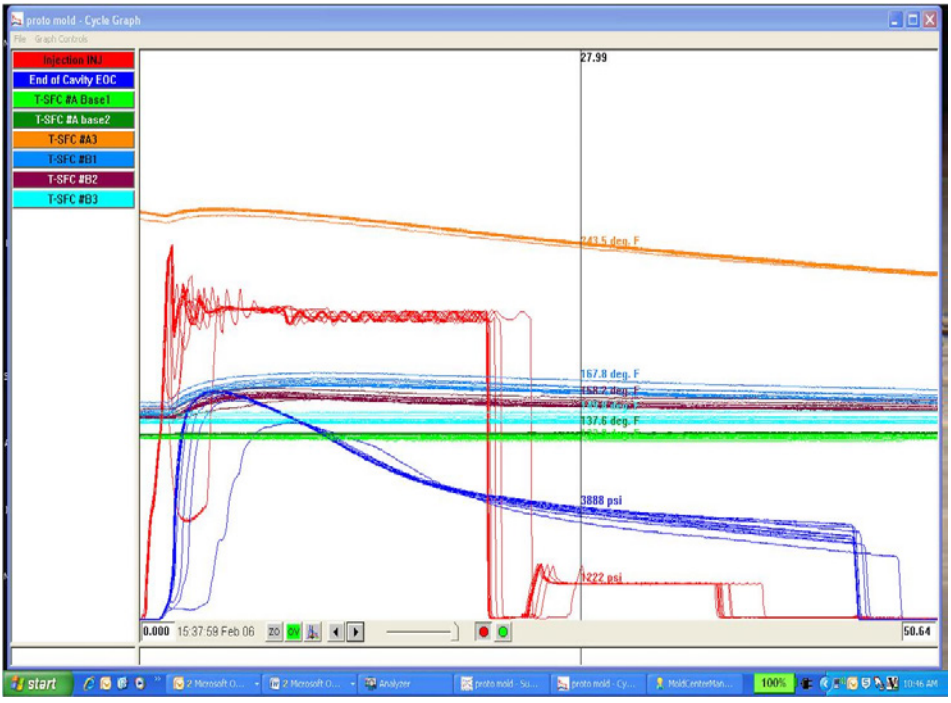


Figure 8.8 eDART® data curves for a machine with terrible 10-shot repeatability; this simple test shows a press that is going to produce large amounts of variation in part quality

This method will show a wide variety of odd machine characteristics that may need to be dealt with from a maintenance standpoint. Figure 8.9 shows a machine with terrible second-stage pressure tuning. Greater access to data will help understand if a machine is performing reliably.

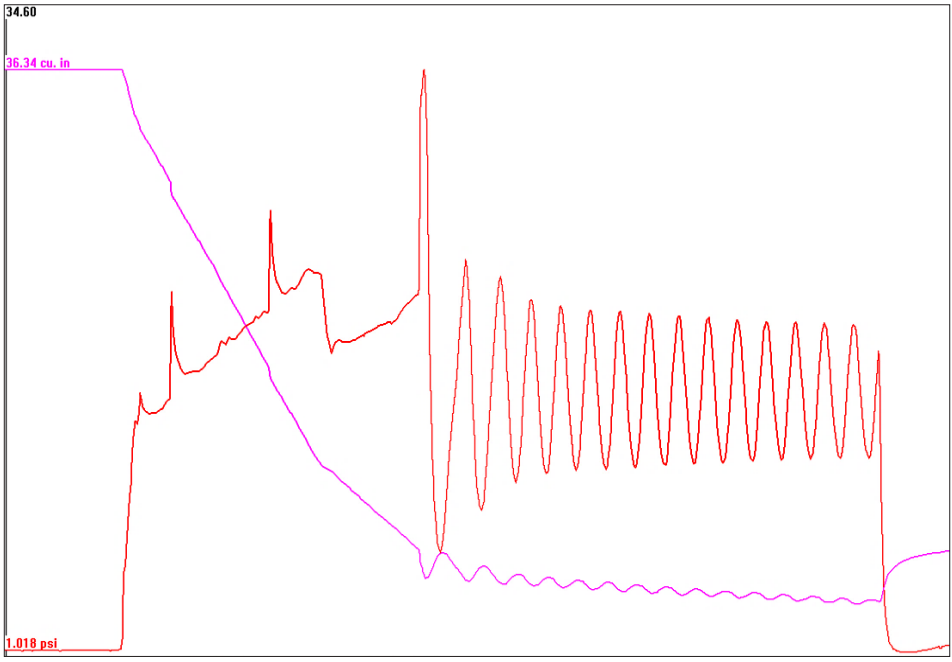


Figure 8.9 RJG eDART® data showing oscillation of hold pressure

8.5 Cycle Variation

Letting the process run in a steady-state operation for a given period of time gives good insight into the consistency of the molding machine. Many factors can be checked for variation over time including:

- Fill time
- Shot size
- Transfer position
- Second-stage pressure
- Clamp open position
- Eject forward position (critical for robotic part removal)

- Recovery time
- Any others desired

All of these process points should be consistent over one or 24 hours. For all of the above other than recovery time the recommended variation should be within 1%, recovery time allows for more variation (5%) [1]. Variation in any of the key process parameters means that the molded parts will vary over time resulting in unacceptable parts. See Figure 8.10 for an example of hold pressure variation which led to variation in part dimensions.

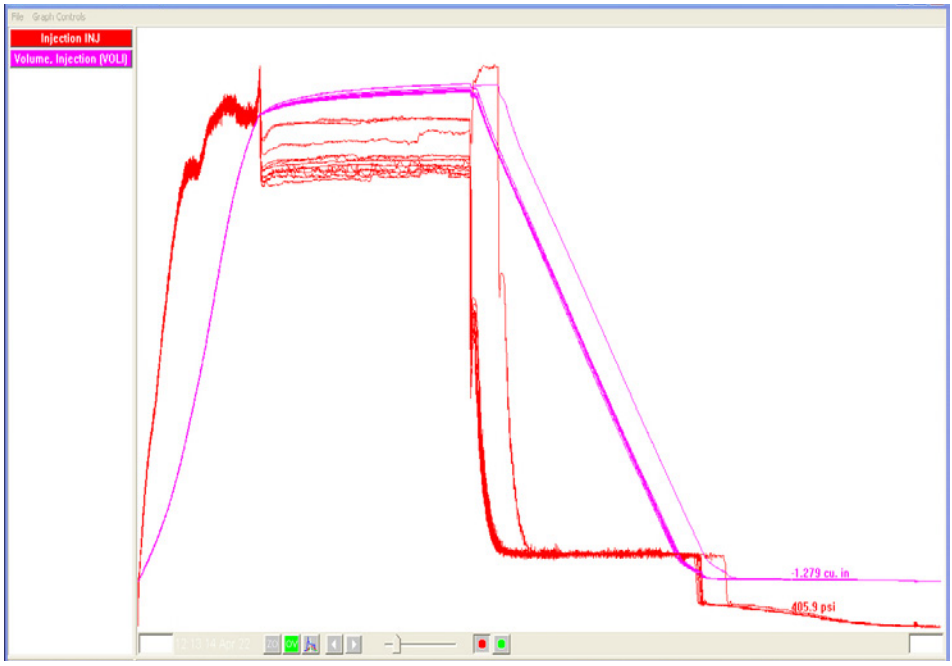


Figure 8.10 Hold variation over 10 shots

There are various ways to check for longer-term machine consistency the easiest of which is to track the machine actual values for specified variables. The second method to capture long-term machine data is to use a process monitoring system to record the data.

8.6 Velocity to Pressure Transfer

A key part of the injection molding cycle that is often overlooked is what occurs when the machine hits the transfer position. At transfer the machine is changing from controlling velocity with an abundance of pressure to controlling pressure. How well the machine makes this transition is critical to part quality. Many defects can occur due to a poor velocity to pressure transfer including:

- Sink
- Short shots
- Flash
- Gloss variation
- Sticking
- Dimensional issues
- Pin push

The factors that influence velocity to pressure transition the most are:

- Fill only weight:

  If the cavity is not filled far enough during first stage the machine will not be able to reach the set hold pressure effectively. In this case the machine will typically have a period of undershooting the setpoint, and then overshoot the setpoint before finally settling in to the set second-stage pressure.

- Second-stage volume settings:

  If the volume setting is not providing enough flow to achieve set pressure, whether it is called hold, pack, or second-stage volume or speed, it is having the same impact. Note that this is not a setting where one value works for all molds. See Figure 8.11 and Figure 8.12 for examples of good and bad velocity to pressure transfer.

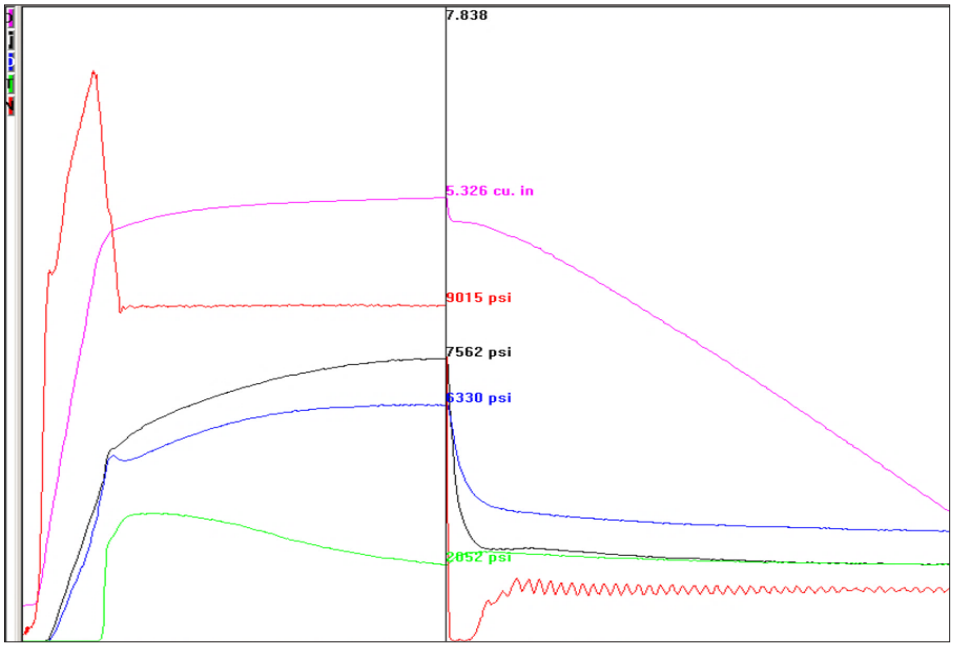


Figure 8.11 Velocity to pressure response with cavity pressure: normal response

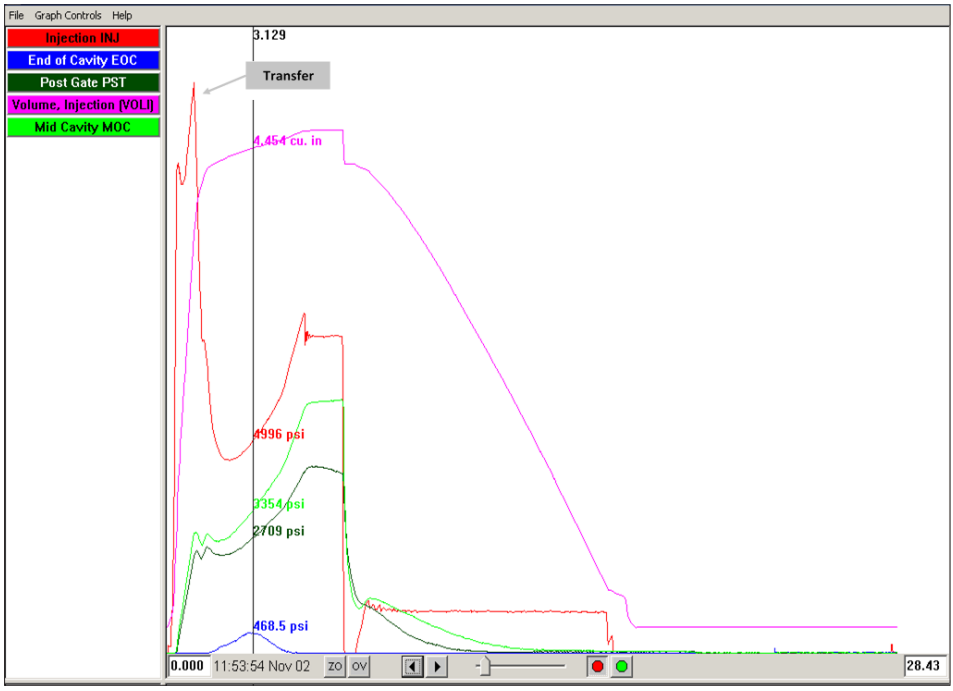


Figure 8.12 RJG eDART® graph showing the impact of a poor velocity to pressure transfer. Notice how all of the plotted lines drop down after transfer. This created dimensionally unacceptable parts.

- A machine can also have issues with the ability of the valve to respond as the machine transfers from velocity control to pressure control. Tuning issues for the valve can impact the ability of the machine to react. There will be cases where the back pages of the machine controller must have adjustments made to ensure that the valve is responding adequately.

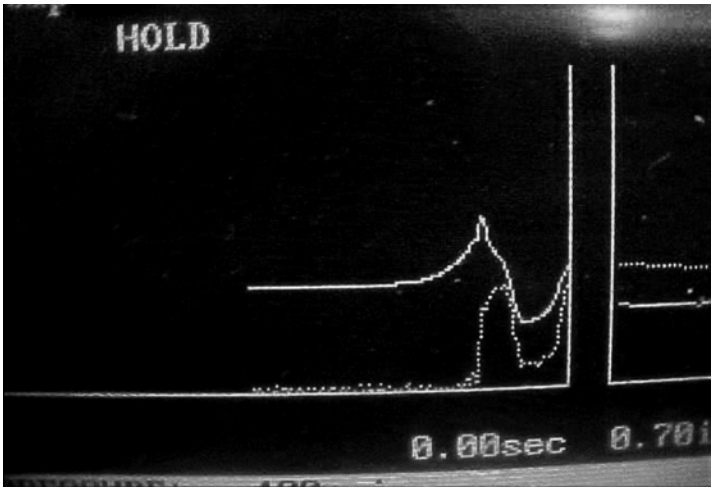


Figure 8.13 Poor velocity to pressure response machine control screen; notice the dip and spike in pressure.

This issue is not isolated to just hydraulic response. Electric presses will have a velocity setting for the second stage that will also have a significant impact on the process (see Figure 8.14). This RJG eDART® screen shows the machine pressure, velocity and cavity pressure for three cycles. The three different sets of curves are for three different set hold velocities. Note that on this process transfer actually occurs at the bottom of the machine pressure curve not at the peak point.

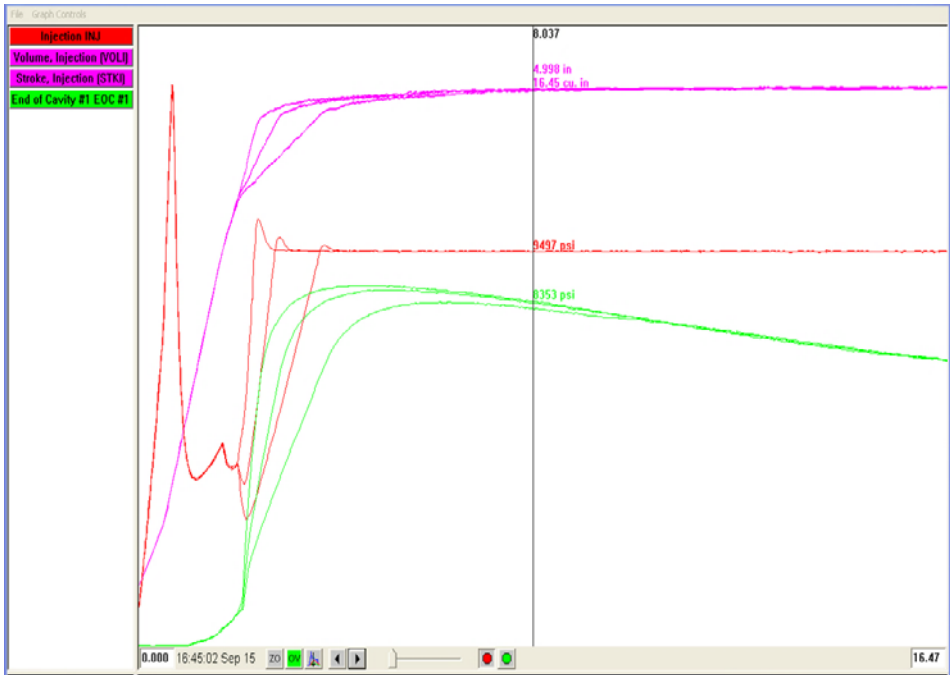


Figure 8.14 Velocity to pressure response on electric press. Each of the three curves has three cycles and the quickest response on the curves was with the fastest hold velocity setting.

8.6.1 Machine Control

Injection molders should be in the habit of verifying if the machine is "doing what we ask, when we ask." Do not assume that because a value was set on a controller that the machine is actually achieving the value. Machines can become out of calibration on a wide number of crucial settings. If the machine is not achieving the desired setpoint the result can be a wide range of defects.

With the wide variety of machine controllers, it is a challenge to have every processor in the shop understand how each controller performs. Controllers will have a variety of process settings to achieve the same results. For example, first stage may be called fill, boost, injection, or velocity and may have units of in/sec, mm/sec, cm/sec, in^3/sec, cm^3/sec, or %. Figure 8.15 shows a variety of machine controllers; does everyone who uses the various controllers know how to use them?

Figure 8.15 Does everyone understand how each controller works?

Key items to check include the following

- Pressure:

  Does the machine setpoint match the hydraulic gauge and the displayed pressure on the controller? If not, which one is correct? Use a secondary hydraulic pressure gauge or a process monitoring system with a hydraulic pressure transducer to measure the actual value. The gauge on the machine may need to be replaced, or the pressure output calibrated.

  Keep in mind that if the pressure is not accurate second-stage pressure can create defects as can back pressure. Pressure should be evaluated throughout the cycle to determine where variations or accuracy problems are occurring. Back pressure can result in variation to the melt temperature and viscosity of the material, which can than impact much of the process.

- Temperature:

  Are all of the heater zones on the barrel and nozzle working? Look for zones that are not achieving the required temperature because that is a likely indicator that the machine may be experiencing heating problems. Also examine the current draw on the various zones for any zones that are calling for either 100% or 0% output. A properly working heater zone will cycle on and off as the zone heats up. Individual heater bands can be checked with an “Amp Clamp” type of tester to verify if the band is drawing current; if not the band may be burned out. Another problem that can occur is when two thermocouples get swapped on zones,

and as the zones are heating the thermocouple will not respond, which will cause the controller to keep sending current to the zone. Figure 8.16 shows a nozzle with failed heater bands captured in an infrared (IR) image.

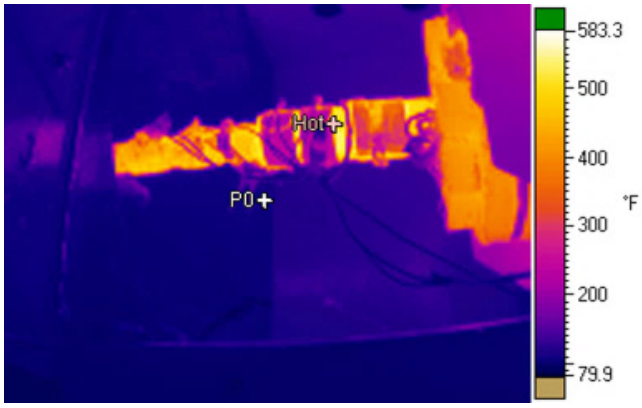


Figure 8.16 IR thermal image of extended nozzle with heater bands not working.

- Examine thermocouples for full insertion into thermocouple wells. Also verify that heat bands are tight against the barrel. Loose heater bands will have an air gap that will restrict heat transfer. See Figure 8.17.

Figure 8.17 Nozzle cross section showing fully seated thermocouple.

- If a heater zone is overriding the setpoint there may be a PID tuning issue with the machine or excess shear may be overheating the barrel. If a zone is overriding the zone is not being controlled by the machine. Try running a PID tuning cycle on the zone, and if it does not help it may be necessary to raise the barrel temperature setpoint to allow the machine to control the zone.
- Other potential concerns relate to thermocouples. If the wrong type of thermocouple (Type J instead of Type K or opposite) is used the temperatures will not be read accurately. Table 8.1 shows the color coding of wires on the typical thermocouples used in injection molding.

Table 8.1 Color Coding of the Most Common Molding Thermocouples

	Minus (−)	Plus (+)
Type J	red wire	white wire
Type K	red wire	yellow wire

- Positional control: How well a machine hits position setpoints need to be watched over time. If a machine is showing variation in shot size or transfer position the fill only weight injected during first-stage fill will be impacted.

  If a machine does not open to the same mold open position robot handling will be compromised. If frequent adjustments are being made to the clamp or the robot it is likely that the mold open position is varying. Ensure that the clamp has an adequate clamp open slow down setting; the machine may be trying to stop too abruptly.

  Verify repeatability for the ejector stroke, as this will also impact the robot handling of the parts. Also, if the ejector stroke is varying on a mold that needs full stroke distance to clear the lifters there will be issues with the part sticking on the lifters.

Remember that the molding machine should “do what we want when we want” shot after shot. Machine performance is one of the keys to successful molding operations. Without a picture of how the machine is reacting troubleshooting will be much more challenging and a feeling of working in the dark can be the result.

8.7 Machine Documentation

It is important to maintain reliable records for all molding machines. Key items that need to be known include:

- Platen size
- Ejector pattern

- Distance between tie bars
- Screw type
- Screw L/D
- Screw compression ratio
- Screw diameter
- Intensification ratio
- Tonnage
- Shot size
- Maximum fill rate
- Maximum injection pressure

For ease of access create plaques that can be mounted on the machine to provide key data. Figure 8.18 is an example of machine data that can be posted on a molding machine. A key to people using machine data is making it readily accessible.

Machine #15	
Screw diameter	80mm
Peak pressure	31,000psi
Shot size (GPPS)	51 oz
Screw type	Mixing
Ri	15.5:1
L/D	20:1
Peak flow rate	44 in3/sec

Figure 8.18 Example of a machine data plaque

Repeatable molded parts need to be produced with a repeatable process and a repeatable process requires a repeatable machine. Avoid processing around machine problems or otherwise the process window will be reduced, which will result in scrap and PPM problems.

If a mold is to be validated in multiple machines the lowest common denominator will be what the process should be established at. For example, if a mold is validated in a primary and secondary machine the first-stage speed cannot be set faster than both machines can obtain. Clear documentation of machine details provides for quick analysis of whether a job will run in a given machine.

References

[1] Bozzelli J., Groleau R., and Ward N. "The Machine Audit: A Systematic Evaluation of Injection Molding Machines: How to Tell a Good Machine—Old or New" Antec 1993.

[2] Doyle K., "Know Your Machine", *Plastics Technology*, May 2014.

[3] Bozzelli J., "Know the Basics of Machine Evaluation, Part 1", *Plastics Technology*, June 2010.

9 Drying

9.1 Introduction

Removal of moisture prior to processing is absolutely critical for molding success. Moisture in plastic pellets will turn to gas when subjected to molding temperatures. This gas will be contained in the plastic melt until the plastic enters the mold where the depressurization on the melt stream will allow the gas bubbles to reach the surface of the mold, leaving behind the streak of splay. Hygroscopic materials such as ABS, polycarbonate, nylon, TPU, polyesters, cellulosics, or PC/ABS absorb moisture from the environment and require drying.

Some additives including fillers and impact modifiers can result in non-hygroscopic materials needing to be dried. There will be cases when non-hygroscopic materials are literally soaking wet (for example from a roof leak), and in these cases the material will need to be dried prior to molding.

Material suppliers will provide a recommended drying temperature and time for hygroscopic materials. It is critical to follow these drying specifications to ensure that the material is dry enough to successfully process (see below for drying requirements).

9.2 Keys to Drying

Successful drying requires the following:

- Correct temperature
- Dry air
- Air flow
- Time under the above conditions

To ensure adequate drying it is critical to have all four of these conditions met. Four hours of drying time is meaningless if the temperature requirement is not met. A typical desiccant dryer will provide an air dew point of −40 °F. Keep in mind that the dryer temperature will help release the moisture from the pellets, the low dew point will allow the air to pick up the moisture, and the air flow exposes more of the pellets to the warm/dry air. Figure 9.1 shows an example of a typical drying hopper.

Figure 9.1 Typical drying hopper

There are many ways that incomplete drying can occur that relate back to the four key drying parameters.

9.2.1 Temperature

Too low a temperature can come from the following problems:

- Drying temperature set too low:

  It is vital to follow the material suppliers recommended dryer temperature settings. If the temperature is set too low the moisture will not be released by the plastic resulting in lack of drying.
- Incorrect location of dryer control thermocouple or RTD:

  The temperature should be measured at the hopper inlet. If the temperature is measured at the dryer outlet there will be a drop in temperature before the air reaches the material. The temperature set point must account for this temperature drop if you are controlling based on dryer outlet rather than hopper inlet. For improved efficiency use insulated hoses between the dryer outlet and the hopper inlet as this will limit the amount of heat loss.
- A burned-out heater can prevent a dryer from achieving the set process temperature. If the dryer is alarming for low temperature the heater may need to be checked and possibly replaced.

Bear in mind that a higher drying temperature is not the route to faster drying! If a material is dried at too high of a temperature it will become tacky or even melt, which will result in what is often called a “hard ball”, which is when the material pellets stick together and will not feed through the drying hopper. A “hard balled” or “rocked” drying hopper means hours’ worth of difficult work trying to remove the stuck plastic. This is an experience that creates a wonderful learning opportunity, and the person who set the temperature too high should be the one who has to remove the melted plastic; chances are they will not repeat this mistake!

9.2.2 Dry Air

As the moisture is released from the material by heating the moisture must be carried away. Moving dry air through the drying hopper will allow the moisture to be carried away from the plastic. Without providing the dry air the water molecules have nowhere to go and as a result the material will stay wet.

The dryness of air is measured by its dew point, the temperature at which moisture in the air will condense. The dew point of the air should be between −20 and −40 °F for effective drying. Common reasons for not reaching the required dew point include:

- Bad desiccant, either due to the age of the desiccant or desiccant that has been contaminated with plastic fines or byproducts not filtered out of return air. As desiccant pellets can and will go bad, use a dew point meter to determine if there

are issues achieving a low enough dew point. Some dryers have dew point meters built into them or portable units can be utilized to monitor a dryer.

- On dryers with multiple desiccant beds you must verify the dew point from each of the beds, and you may find one bad and one good desiccant bed. This means that you will need to verify the dew point over a time period. One way to verify the dew point over time is to purchase a chart recorder to connect to the dew point meter. Another method is to connect the dew point meter to a data monitoring system such as RJG eDART®.
- Return air too hot:

  For optimal performance of a desiccant the return air should be between 120 and 150 °F. If drying set points are above 180 °F the return air will probably be too high for optimal performance. When drying at temperatures above 180 °F an after cooler should be used to cool the return air to below 150 °F. Also keep in mind that return air hoses should not be insulated as this will allow the return air to cool as it travels back to the dryer.
- Burned out regeneration heaters will not provide enough heat to remove the absorbed water from the desiccant. If the regeneration heaters are not working you will see a high dew point.
- Make sure that there are no leaking seals or holes in the drying hoses that would allow moist ambient air to be introduced into the drying hopper.

How to Use an RJG eDART® as a Dryer Monitor

Connect the output from the dew point meter to a 0–10 V analog input module. You can also use a dew point meter from RJG that will connect directly to the eDART® without the analog input module.

Anywhere that you wish to collect temperature data from can have a thermocouple installed, and the thermocouples can then be connected to an RJG Quad Temp Module. You could measure hopper inlet temperature, dryer outlet temperature, and maybe even regeneration temperature.

With the above information you can establish full time monitoring with a permanent eDART® or setup a portable dryer qualification methodology where you monitor the dryer for 24 hours to determine how well it is working.

Checking Desiccant

To verify if a desiccant is working conduct the following experiment: Dry desiccant can be taken from a dryer desiccant canister or it can be dried in an oven for 2 hours at 400 °F (placed in an appropriate container). Allow the desiccant to cool to room temperature and then pour some water into the container with the desiccant. If the desiccant is active there will be a violent exothermic reaction (use caution!) as the desiccant absorbs the water, steam will be observed, and a significant temperature rise will be detected (> 20 °F). If the desiccant is not active there will be no reaction or temperature rise.

An inactive desiccant should be replaced in the dryer because it will not obtain an adequate dew point with a bad desiccant.

9.2.3 Air Flow

Poor air flow can come from the following issues:

- Plugged filters restricting the air flow:

  All dryer filters must be kept clean. Do not run dryers without filters or the desiccant bed will become contaminated and not be capable of achieving adequate dew points.
- Feed hoses can become crushed, which restricts the air flow. Verify that all hoses are free of crushed areas and holes.
- Too small of a dryer for the drying hopper:

  Dryers are measured in cubic feet per minute (CFM) of airflow. If the dryer is undersized relative to the hopper there will not be enough air movement in the hopper to effectively reach much of the material.
- A burned-out blower will result in no air flow. Check that the blower is running and has not burned out. Verify that the dryer is not wired with reverse polarity or the blower will run backwards.
- Most modern dryers will alarm if the airflow is inadequate. Do not make a habit out of silencing alarms on equipment: It is ringing to tell you something.

9.2.4 Time

Lack of drying time typically comes from the following:

- Simply starting up the machine prior to the required amount of drying time:

  This is a plant discipline issue, and processors must know that the material has had adequate residence before starting the press.
- Allowing hoppers to run down before filling them:

  If they run too low you will have material that has not dried long enough. This is also a plant discipline issue. Material handlers cannot be allowed to let hopper volumes run down.
- Material flow:

  Molding expert John Bozzelli has conducted studies that show that some dryer designs will tend to have a “rat hole” flow where the center pellets travel much faster than the outer pellets in the hopper [1]. This is a hopper design problem

that impacts material residence time in the hopper. This study can be replicated by filling a hopper and adding a layer of another color of pellets to the top of the dryer as tracer pellets. Start loading the material from the hopper and note how long it takes to see the alternate colored pellets.

- Bear in mind that the material supplier recommended drying times may not account for material that has become very wet due to sitting in open containers in a high-humidity environment. Additional time may be required to ensure that the material is dried to an adequate level.

Note that too much time spent drying can have a negative impact on some materials. What is normally considered an over-drying problem is a lack of moisture content in a material like nylon that in turn leads to a higher material viscosity (the water was acting as a plasticizer). When nylon is dried to very low levels of moisture content the viscosity change may actually lead to a pressure-limited condition that impacts the ability to fill the mold. There are cases when materials are dried much longer than the recommended drying time so that oxidation of the material can occur, which can lead to a breakdown of the physical properties of the material. Always try to avoid leaving material at drying temperatures for extended periods of time; dryer set points can be reduced to maintain a material at a dried state without risking over-drying problems.

9.3 Moisture Analysis

Material moisture content can be verified prior to molding by utilizing a moisture analyzer. There are two common types of moisture analyzers which have very different techniques:

1. Carl Fischer titration method:

   This method relies on precise analytical equipment and requires chemicals to run. This test will provide a true moisture reading from a plastic.

2. Loss on weight method:

   This method uses a very precise scale to weigh the material at the start of the test. The material will be heated forcing it to give up moisture and the moisture analyzer will calculate moisture percentage based on this weight loss. Note that this method is more convenient but is less accurate because it will also measure other things that leave the material including residual monomers or low molecular weight additives.

Every plant must evaluate their choice of moisture analyzer. Carl Fischer titration will provide more accurate results but the cost of that is measured by the fact that it is a lab-style piece of equipment that takes more skill to run and maintain.

Figure 9.2 shows an example of a typical loss on weight moisture analyzer and Figure 9.3 shows an example of the typical printout for a moisture analyzer showing dry material.

Figure 9.2 Loss on weight moisture analyzer

Figure 9.3 Printout from moisture analyzer verifying material is dry

9.4 Material Handling

It is important to understand that just because a material has been dried to an adequate level does not mean it will stay there. If you expose dried material to ambient air that has humidity in it the material will begin to absorb the moisture like a sponge. Many materials will be too wet to effectively process if exposed to ambient humidity for longer than 15 minutes. To avoid this creating splay problems, ensure that minimal material is allowed to be outside the drying hopper as it is transported to the press. This can be accomplished in a couple of different ways including:

- Use of small one-shot hoppers on the molding machine (see Figure 9.4):

  This will keep the volume of material outside of the drying hopper minimized. Set the proximity switch on the one-shot hopper according to the required shot volume.

Figure 9.4 One-shot hopper

- Placement of the drying hopper on the feed throat of the machine (see Figure 9.5):

  This will eliminate any issues with external transfer of the material from the drying hopper to the machine hopper. This works reasonably well with small hoppers but as they become larger cleaning them out during material changes can be more challenging.

Figure 9.5 Drying hopper mounted on a molding machine

When dealing with materials that are extremely hygroscopic remember that how they are stored will impact drying time. Nylon stored in an open container in hot humid environments may require 10 hours or more to fully dry as compared to the 2–4 hours drying required for material from a fresh foil sealed bag. Try to keep the opening for the material loading tube as small as possible and when finished roll the bag up to try to seal against the environment.

Never forget that the drying time recommended by the material supplier accounts for average values. When drying very wet material the extra required time must be accommodated prior to molding startup.

9.5 Material Concerns

As mentioned in Section 9.1 the type of material has a dramatic impact on the drying process. Every material has a moisture absorption rate, a maximum moisture content, and a specific required temperature to achieve optimum drying.

It is important to understand that the additives in a non-hygroscopic material can absorb moisture and in turn require drying prior to molding. Many mineral-filled polypropylenes require 2 hours of drying to remove the surface moisture from the filler content.

For materials that are extremely moisture sensitive the time that the material can be exposed to humid air can be very short. TPU is one of the worst, and studies have shown that within 5 minutes of leaving a drying hopper TPU can actually absorb enough moisture to make it too wet to mold successfully. Take care to minimize the time materials spend outside the drying hopper.

Do not forget that splay is not the only result of molding wet material. Some materials will experience hydrolysis when molded with excess moisture content, which will lead to reduction of physical properties that may cause part failure. Also understand that some materials such as TPU or polyesters will become sticky and weakened when molded wet. When molding wet TPU or polyesters the first sign of problems may be a part stuck to the mold, or cracking parts during ejection from the mold.

It is also vital to understand that when a plastic suffers from hydrolysis the molecular weight of the material will be reduced. A reduction in molecular weight will lead to reduced physical properties that may compromise the performance of the molded part. Beware that hydrolysis is not a reversible process: re-drying a material that has been processed wet will not put the molecular chains back together. Once a material has undergone hydrolysis from being processed wet it should be scrapped or otherwise it can cause processing and performance problems.

References

[1] Bozzelli J., “Bozzelli’s Guide To Specifying A Dryer”, *Plastics Technology*, April 2014.

10 Purging

10.1 Purging Impact

When a molding machine is changed over from one color or material to another it is critical to do a thorough changeover and purge. If a changeover is not done correctly hours of scrap can be produced. Every molding shop should establish procedures for a proper changeover of materials or colors of material.

Poor changeovers can result in:

- Color streaking
- Contamination
- Delamination
- Splay
- Black/brown streaking
- Black specks
- Cloudiness
- Plugged hot tips

Note that scheduling can be a major help to changeovers. One very helpful hint is to keep like materials and colors grouped in specific presses as much as possible. Downtime can be reduced by grouping materials of similar process temperatures, which reduces waiting for barrels to cool down. Also, if making color changes try to go from light colors to dark whenever possible. The purge in Figure 10.1 shows the transition from black to gray; also note that this machine appears to have had extended downtime prior to purging due to the presence of degraded material.

Figure 10.1 Example of degraded purge

■ 10.2 Purging Method Example

This is an example of a simple step-by-step purging procedure.

1. Prior to shut down, verify that the new material is the correct material, dried, and ready to be used.
2. Shut off material feed based on number of remaining shots to be molded.
3. Drain hopper of any extra material.
4. Back barrel away from mold after completing required quantity of parts.
5. Begin using inject forward and screw rotate to purge the material through the barrel, and purge until barrel is empty. Increase injection velocity to purge some high-velocity shots to try to increase the cleaning action.
6. Clean all hoppers and material feed lines of previous material.
7. If using purging compound begin feeding into the machine at this point. Follow the purging compound recommendations for quantity, back pressure, and any specific methodology.
8. Using screw rotate and inject forward to purge the purging compound completely out of the barrel. Vary the back pressure from low to high as well as the injection speed to increase agitation while purging.

9. Clean all hoppers and feed lines of any purging compound. Be sure to clean off any shelves that can trap pellets, such as the edge of the magnet drawer.
10. Begin feeding new material to the machine.
11. Using screw rotate and inject forward to start purging the new color or material through the barrel. Once any residual color swirls from the purging are gone the press can be restarted.
12. If using a hot runner mold try raising the temperatures 30–50 °F to improve cleaning. Use a hot runner safe purging compound to speed manifold cleanout.
13. Note that nozzles and tips can trap material for many shots and it is often faster to remove the nozzle and tip and clean rather than relying on the purging action.

10.3 Purging Compounds

There are many commercial purging compounds available on the market. Some compounds rely on an abrasive scrubbing action, some rely on chemical reactions, and some use a combination of the two. Experience says that they are well worth evaluating for impact. To calculate the value of using a purging compound try the following:

1. Purge with baseline method. Document the amount of time it took from the last good part to the first good part of the next color. Also document the amount of material used to make the color change. Finally document the scrap encountered during and after the color change.
2. After conducting step one, do the same with the purging compound being evaluated. Document time to changeover, weight of purging compound used, weight of material used, and scrap during and after change.
3. Do the math. Even though a purging compound is expensive it may yield a much cheaper material change.

Because a purging compound is costly, make sure that it is being used correctly. If using a purging compound everyone must be familiar with the correct procedure. If someone purges without using the correct procedure they can waste hundreds of dollars in purging compound, which in turn gives the purging compound a bad name.

11 Hot Runners

11.1 Hot Runner Advantages

Hot runners are used for a variety of reasons and have many more uses today with the advances in technology than they had years ago. They provide the following advantages:

- Are used to eliminate the cold sprue/runner, which can save you money with scrap when regrind is not an option. Of course there is an added cost to the tool up front so justification is always needed with savings to offset the cost.
- Allow direct gating to the part with a low vestige tip or valve gate and eliminate the need for three-plate runner systems.
- Allow gating on the outside edge of the part when a part is centered in the mold.
- On larger parts where knit lines or flow lines are a concern sequential valve gates allow control of the flow and to move the knit lines.
- Can use multiple hot drops to reduce fill pressures.
- Can be used to eliminate long cold runners to reduce the pressure loss and improve the process window.

11.2 Heaters and Thermocouples

Think of a hot runner as an extension of the molding machine itself. Its purpose is to keep the plastic molten at a proper temperature on its path to the cavity. The steel that contains the flow channels needs properly placed heaters and thermocouples to ensure the flow channel is at and maintaining the proper temperature. The thermocouple itself is the most critical component of this process. The thermocouple is what measures the temperature of the component and tells the controller how much current (amps) to send the heater to maintain the set temperature. So in

most cases when there is an issue with burning or inappropriate temperatures the thermocouple is the root cause and not the heater. A heater is basically like a light bulb: it is either working or burnt out, but there are times when it appears it is functioning but is not able to come up to temperature. If a thermocouple is loose or not held tight against the steel it will be reading the ambient air temperature causing the controller to send excessive current to the heater and causing overheating.

Overheating can also occur when the thermocouple and heaters are not wired to the correct zones on the connection plugs. Modern hot runner controllers have the ability to determine if hot runner heater and thermocouple zones are correctly matched.

11.3 Hang Up Areas

With today's hot runner designs hang up areas in the flow channel are not an issue in most cases. The one area on the hot runner that causes the most issues leading to cosmetic defects on the part is the hot drop tip, which depends on the tip style and insulator caps available. The other area to consider is the orifice on the hot runner and the orifice on the nozzle tip; they should be one to one in size to prevent hang up of material.

11.4 Hot Drop Tips

There are three general styles of hot drop tip designs with many variations of each design. The three types are (1) the low vestige tip, (2) straight through sprue style, and (3) the valve gate.

1. The low vestige tip is used to direct the gate onto the part with minimal vestige and also used on cold runners to reduce stringers. These can increase fill pressures (pressure loss) and contribute to color issues when running multiple colors. They will also plug more easily with the orifice having less volume/area.
2. The straight through style tip is usually the least problematic with color changes and contamination because there are fewer areas for the material to hang up. But even with some of the new designs considering the insulator gap even a sprue style can create problems. Most systems provide components to fill these gaps commonly known as color seals.

3. Valve gates are used to direct the gate onto the part with minimal vestige and are also used to control the flow when using multiple valve gates. Because they can be shut off independently you can control flow fronts and knit line. When doing a color change with a valve gate the previous color may stick to the pin and continue to drag out. Cycling the valve open and closed can help with breaking free of the previous material or color.

11.5 Drooling, Stringing, and Sprue Sticking

Drooling from the hot drop tip or sprue is usually a result of a lack of cooling or lack of bearing surface. Tip designs over the years have incorporated insulator gaps and minimal contact with the tip to prevent freeze off and heat transfer to the cavity blocks. In many cases and with some materials this is a good thing. But with some materials an increased contact surface with the cavity steel along with cooling lines around the hot drop is necessary to prevent drooling, stringing, and sprues from sticking.

11.6 Freeze Off

Freeze off in the tip area is usually caused by a lack of heat or the orifice size being too small. With some materials, especially semi-crystalline polymers such as nylon, temperature control at the tip is critical. This is when you want minimal contact surface with the tip and the cavity steel. Many hot runner suppliers use different tip designs to increase temperature control at the orifice. Low vestige tips have a pointed insert, typically called a spreader tip, that is designed to maintain temperature control at the orifice to prevent freeze off. The location of this spreader tip in relation to the orifice is critical and manufacturer specifications should be followed.

11.7 Orifice Size

The orifice size will depend on the material, wall stock, and tip style being used. It is important to keep an open mind with orifice sizes when addressing issues with pressures, high gates, drooling, freeze off, and scrap.

Selection of proper orifice size must be analyzed carefully to ensure a balance between all factors. Computer-aided flow analysis can help with predictions of how well a given gate orifice will be at filling and packing a mold. Consult with material suppliers and hot runner manufacturers for recommendations on orifice sizes.

Case Study

This concerns a part having a scrap issue with splay where the orifice size ended up being the solution. This was a part with four low vestige tips that had a spreader tip in the center of the gate orifice. Many will just consider the orifice diameter and not include the area of the spreader tip that reduces the volume. In this case the orifice diameter was 0.050″ and the spreader tip in the center of the gate was 0.025″ diameter. The area of the spreader tip reduced the area/volume by 25%. The fill speeds with this gate needed to be on the high end to make a good part but we struggled with a lot of splay. We opened the orifice diameter to 0.060″, which was an increase of 55% in area/volume when considering the area that the spreader tip took away. This allowed us to adjust the fill speed and eliminate our splay/shear issue.

11.8 Leakage

Hot runner leakage can be a major problem that will shut down a mold. A big development toward reducing hot runner leakage occurred when the hot drops started being threaded to the manifold versus relying on the stack height, seal rings, and bolt patterns in the mold plates to hold the hot runner together. Some toolmakers complained from a maintenance perspective that this was more difficult to work on when the drops had to be removed for service. In some cases this was true with threads getting galled up and creating another mess. But improvements with thread designs and coatings have reduced this concern among toolmakers. So in the big picture of hot runner issues this has been a big improvement for maintaining molds.

Another observation over the years, but not a common one recently, is with support in the hot half. If the areas cleared out for the hot runner are excessive, cavity pressure can lead to deflection within the mold plates leading to leakage.

11.9 Zones and Wiring

Hot runners have zones, typically identified with numbers, which are dependently controlled with a heater and a thermocouple. It is important to have the wires labeled for each zone so it is easier to troubleshoot versus having to chase wires down, especially when multiple zones are used. A hot runner schematic should be on the side of the tool showing each zone, location, and the heater watts.

The power plug/connection and the thermocouple plug/connection are also identified by zones. So, for example, the zone 1 heater leads would be to zone 1 on the power plug and the thermocouple to zone 1 on the thermocouple plug. The two wires coming off the heater are the same but on the thermocouple they are not. The thermocouple has a positive and negative wire that must be wired to the correct location on the plug or it will not function properly. Similar to the battery in your car, it is not complicated but will not work if wired in reverse. Also, thermocouples come in different types but the J type with the red and white wires is most common. The positive wire is magnetic, and is the white wire for the typical J type, and the red one is the negative. If you are ever in doubt which one is the positive wire you can use a magnet to find out. Make sure to use the proper thermocouple for the chosen hot runner controller. With thermocouples be cautious adding wire extensions unless you are experienced with proper methods and problems with cold junctions. With the heater and thermocouple any connections with extensions must be insulated.

11.10 Hot Runner Troubleshooting

There are some things you can do in the molding machine to troubleshoot hot runners instead of pulling the mold for repair. A first thing to try when having hot runner issues is to change out the cables and controller. Long experience has shown that the controller or cables are often the source of the issue. Also, just because the controller shows a zone at the proper set temperature does not necessarily mean that this is the case if you are having an issue. Also verify the pins in the plugs, because at times they can get pushed in not making contact with the connection on the cable plug.

Use a pyrometer with a 0.040″–0.060″ diameter sheath thermocouple to verify temperatures inside the hot drops or the inlet channel. With valve gates you pull the valve pin back to get inside the drop flow channel (note: always use proper safety precautions including face shields and move the injection unit away from the mold to avoid a blowout while probing tips). Make sure to contact the steel in

the flow channel. Gently move the thermocouple back and forth against the steel to get an accurate temperature. You will get a false reading if you just probe inside the drop because the plastic will form a skin and affect the temperature reading. If you find you have a thermocouple issue and are not getting the proper set temperature you have the option of running on percentage with most controllers. If you have a matching zone with the same wattage heater you can set the percentage until it matches the same current (amp) draw of the matching zone. You must also consider that the amp draw will change as you are running parts because the hot plastic running through the hot runner will reduce the current needed with the plastic releasing heat so the heater will not have to work as hard. If you have a zone hotter than the set temperature on the controller it is more than likely a thermocouple issue. If the thermocouple is loose or not making steel contact with the hot runner or hot drop it will be reading the air temperature, which would be cooler than the steel of the hot runner. It would thus tell the controller to supply more current to the heater, which would then lead to a much higher temperature than the set temperature on the controller.

Heaters are in most cases like a light bulb, either good or bad. But in rare situations if there are damaged wires where some of the strands are broken a heater may not be able to manage all the current required to bring the heater up to temperature. Heaters can be checked for proper function with resistance using a multimeter in the ohms setting. If you know the voltage and the watts on the heater you can calculate what ohms that heater should be at. Take the volts squared divided by the watts to figure out the ohms. For example, if you have a 240-volt heater with 1000 watts the ohms should read 57.6 (i.e., $240 \times 240/1000$).

Water leaks where water gets down into the hot runner system can also cause issues. With most hot runner designs today these do not cause major issues with blown heaters like they would with older designs. In most cases you just need to drain the water out, and dry the system out, and put it back into operation. One sign of a water leak is with zones struggling to get up to, or not achieving, the set temperature.

12 Cavity Balance

Cavity balance is the measure of how evenly a multicavity mold fills. Figure 12.1 shows four cavities from an eight-cavity mold with imbalance, and Figure 12.2 shows the Moldflow predicted fill pattern. It is critical that in all phases of the molding process (first stage, second stage, cooling) all cavities are treated the same. Many defects can be directly linked to poor cavity balance including:

- Short shots
- Flash
- Sink
- Burns
- Voids
- Dimensional issues
- Warp
- Sticking
- Gloss

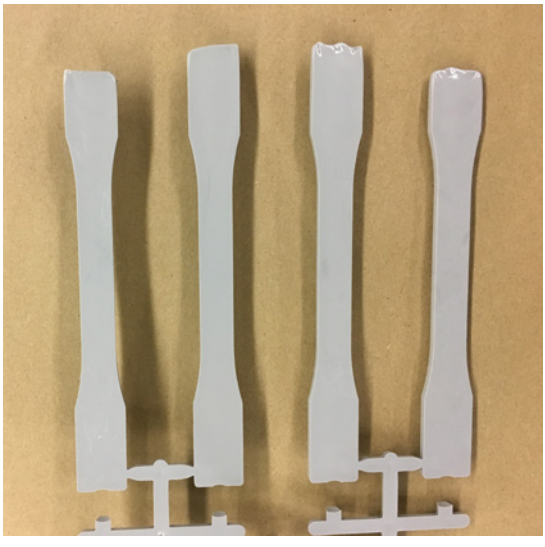


Figure 12.1 Eight cavity fill imbalance

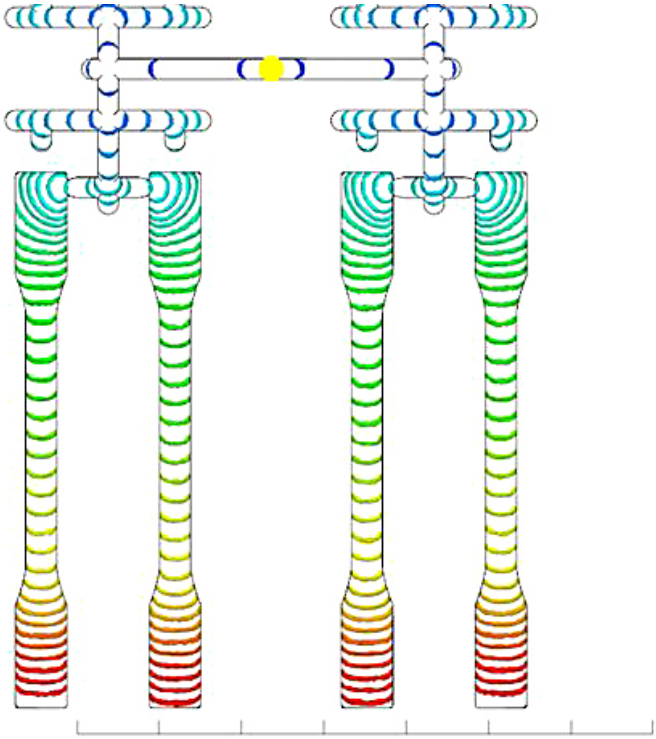


Figure 12.2 Moldflow fill plots for the tensile bar mold in Figure 12.1

A single-cavity mold will always provide the most consistent part; however, it may not be cost effective. Multicavity molds will normally produce parts at lower cost provided that the scrap does not become prohibitive.

During mold design it is critical to ensure that a multicavity will be balanced to provide equal process conditions to each cavity. If a cavity is filling behind the others the process that the plastic experiences in that cavity will be different, which in turn will produce a different part. One of the critical challenges in multicavity molding is to ensure that every cavity is as equal as possible throughout the process. Keep in mind that this must extend to all plastic conditions that each cavity is experiencing (fill, pack, hold, gate seal, shear, cooling, etc.).

Cavities must have the following conditions applied equally:

1. Flow length
2. Flow diameter
3. Shear
4. Cooling
5. Venting
6. Clamp pressure

12.1 Flow Length

A balanced runner is a critical starting point for any multicavity mold. A traditional runner system will have the same linear flow length from the material inlet (sprue or tip) to each cavity. Flow length should intentionally be designed to be identical for all cavities. Figure 12.3 shows a traditional eight-cavity runner system that resulted in unbalanced fill, even though the flow length to each cavity was equal (see Section 12.3).

12.2 Flow Diameter

Just as with flow length it is critical that the plastic flows through the same diameters of runners as it moves from inlet to cavity. Fishbone style runner systems have been built where runner diameters are adjusted based on distance from the inlet. This method of runner layout guarantees that the cavities will experience different conditions.

Flow diameters that have been adjusted to try to fix something can be unbalanced, which leads to additional problems. If the diameter of the runner varies the flow channel also varies, which leads to variation in pressure drop, shear rate, and cooling effects. These differences will result in differences in cavity fill rates, cavity pressure, and cavity cooling. Always ensure that runners have matching diameters for all of the flow paths. A four-cavity mold has four flow paths and all four of them should have equal diameters at comparable locations for each cavity's runner.

12.3 Shear

Even if a runner system is naturally balanced for flow length and diameter, molds with eight or more cavities will still be unbalanced. Everyone who has been around molding long enough has heard a variety of possible explanations as to why an eight-cavity mold is running with imbalance even though the flow lengths and diameters are balanced. The four inner cavities on eight cavity molds will normally fill before the four outside cavities. This effect has been blamed on mold deflection in the center of the mold, increased melt or mold temperature near the center of the mold, or platen deflection, among other possibilities.

In the early 1990s John Beaumont discovered the true root cause of imbalance on molds with multiple cavities [1]. His experiments led him to the understanding that as melt streams are split while travelling through a runner system the high shear layers of material will be consistently positioned in a mold; this is due to laminar flow of plastic within the runners (see Figure 12.4). The Beaumont Effect on mold filling has been solved by John Beaumont's company Beaumont Inc. They have developed a patented technology called the MeltFlipper®. The basic premise of the MeltFlipper® technology is to cause a rotation of the melt stream as it travels through the runner system, which in turn allows an equal distribution of high- and low-shear material to all cavities. For more on the MeltFlipper® contact Beaumont Inc. [2]. The definitive explanation of shear imbalance can be found in John Beaumont's book "Runner and Gating Design Handbook" published by Hanser.

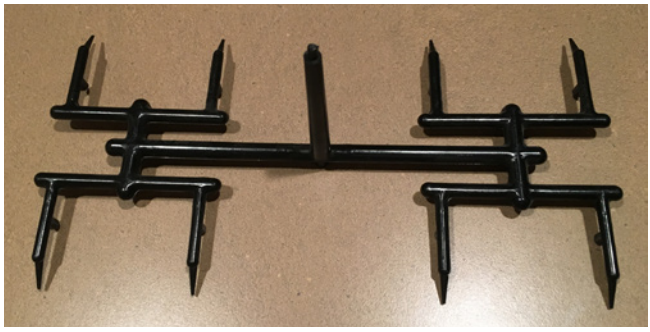


Figure 12.3 Traditional eight-cavity runner that results in an unbalanced fill pattern

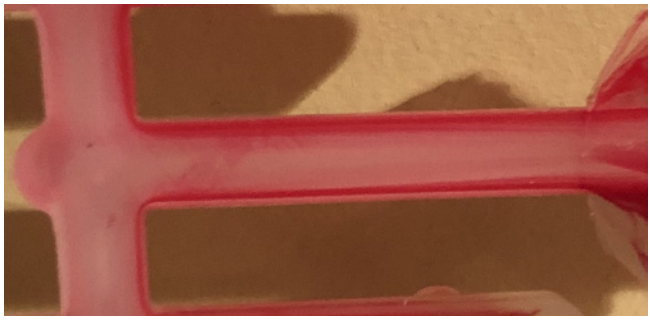


Figure 12.4 Eight-cavity runner system, notice laminar flow of dark tracer flow

Many experimental runner designs have been tried to balance out shear imbalance. If the mold has room a spoke layout (Figure 12.5) is an excellent choice for preventing shear imbalance. See Figure 12.6 for an example of an unsuccessful attempt to eliminate the shear variation on an eight-cavity mold.

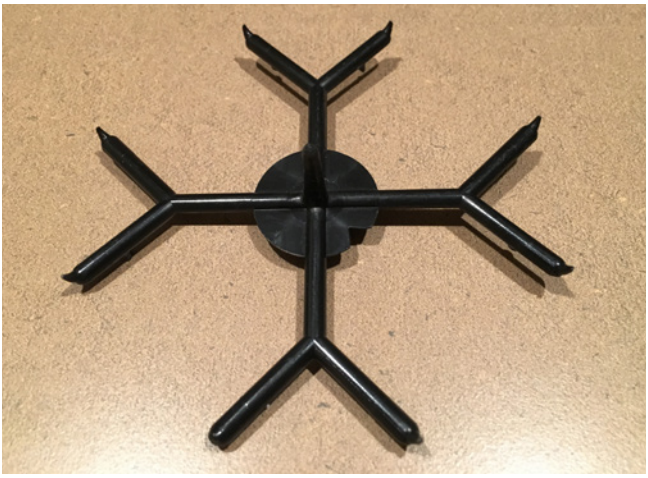


Figure 12.5 Spoke layout runner system for an eight-cavity mold

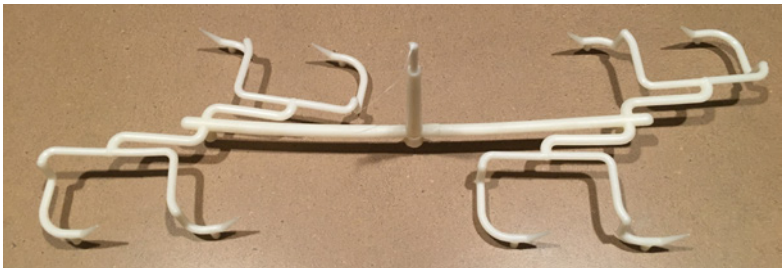


Figure 12.6 Nonworking attempt to minimize shear-induced imbalance

Another option for mold cavity layout that can improve imbalance is to add a hot runner system that allows for groupings of four cavities at a time. In other words, for an eight-cavity mold use a hot runner with two hot drops leading to two individual groups of four cavities. Note that there is evidence that hot runners will still suffer from shear imbalance problems, but it is normally not as severe.

■ 12.4 Cooling

If a mold is not being cooled efficiently cavities may be exposed to differential mold temperatures. This can be a problem that seems to change over time if the mold is not consistently watered. If additional lines are run jumped together the variation in steel temperature can result in part differences. It is critical to ensure that the

mold is watered the same on each run, which is most easily done by hard plumbing jumpers and limiting the ins and outs. A water diagram should be documented for every mold and must be the standard for every mold run. Also beware of swapping a lower flow capacity thermolator; this could lead to insufficient water flow to provide adequate cooling.

Variation in cooling of cavities can be detected by checking the surface temperature with a pyrometer and surface probe. Another means of verifying consistent cooling is to use a thermal imaging camera to take a picture indicating the temperature of the part after ejection.

Some rules of thumb for cooling are:

1. Start the inlet water at the center of the mold where the most thermal load occurs from the hot plastic entering through the sprue.
2. Look for a mold surface temperature variation of 10 °F or less. Document a surface temperature map during process development for future troubleshooting.
3. The difference between the water inlet temperature and the outlet temperature of any circuit should be a maximum of 4 °F [3]. If the difference is larger than 4 °F the circuit should be split into shorter flow lengths or the water flow rate should be increased with a pump of higher capacity.

12.5 Venting

Venting in all cavities must be equal to ensure that they experience the same conditions. If a given cavity is poorly vented compared to other cavities it will tend to burn and/or short. Also examine the runner venting to ensure that the runners are all equally vented; all cavities must be vented the same.

For venting specific details see Chapter 7.

12.6 Clamp Pressure

If the pressure of the clamp is not applied equally across a mold there will be a risk of creating non-uniform venting. This uneven clamp force can come from a number of problems including lack of support in a mold, platen deflection, lack of platen parallelism, or even something as simple as rust buildup on platens. Many of these issues need to be addressed with robust mold design and adequate machine maintenance.

12.7 Cavity Fill Balance Study

Evaluating cavity fill balance is an easy study to conduct. Simply run a fill only shot ensuring that the fullest cavity is still short, and then weigh the parts from all cavities to determine the heaviest and lightest cavities. Cavity fill balance can be expressed by the following equation:

$$\frac{heaviest\ cavity - lightest\ cavity}{heaviest\ cavity} \times 100 = \%\ imbalance \tag{12.1}$$

Depending on the requirements of the parts being molded the industry standard for cavity fill imbalance is less than 3–5%. If cavity imbalance is higher than 3–5% the balance problem must be dealt with to provide consistent parts between all cavities. Figure 12.7 is an example of a spreadsheet form for conducting a cavity balance study.

Cavity	Cavity weight	Balance %
1	112	97.4
2	114	99.1
3	114	99.1
4	115	100.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0
		0.0

Mold % Imbalance: 2.6

((Highest weight - lowest weight)/(highest weight)) X 100

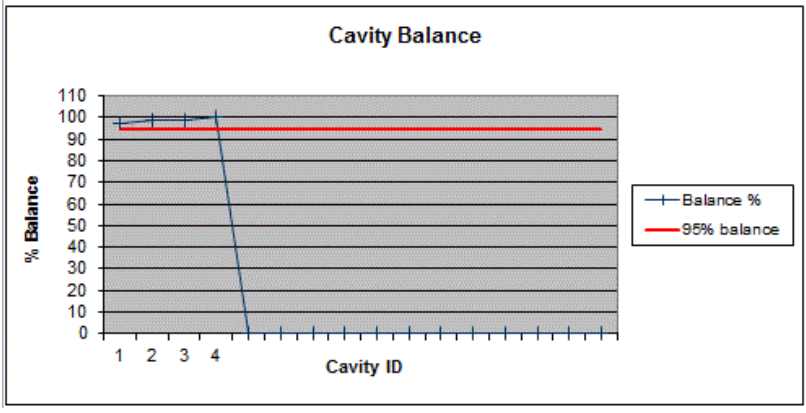


Figure 12.7 Cavity balance worksheet

Cavity balance is also dependent upon the cavities being the same! If the cavities were cut with wall stock variation or other differences in the steel the mold will be unbalanced. Also, a family mold is automatically unbalanced. Family molds will increase all of the challenges experienced in a molding operation and should be avoided.

12.8 Artificial Balance

When molds are experiencing cavity imbalance the obvious desire is to fix the problem so that all cavities are subjected to the same process conditions. One solution that people will sometimes try is a simple fix like adjusting gate sizes or modifying drop temperatures to try to balance the filling of the mold. Unfortunately, by making these adjustments, although the fill only shot may become "balanced", things like pressurization of the cavity and gate seal will be different, and as a result the parts will also be different between the cavities. Note that all gates should be identical in size, taper, and land length.

Case Study: Imbalance from Gate Taper

An important part of the gate design is the mass and taper leading up to the orifice on a sub or cashew gate. If the taper is not identical between cavities the pressure to fill can increase and cause fill imbalances. An example of this was on a two-cavity mold with sub-gates. This mold had a right-hand part and a left-hand part that were exactly mirrored so there was no difference in cavity size but the parts were significantly unbalanced. The sub-gate orifice was opened up on the short cavity even though the gates were the same size (note that gates should be identical). After opening the orifice to the extent it had 50% more volume the parts were still unbalanced; something was obviously wrong! Close examination showed that the taper of the sub-gates was slightly different, but not enough to be noticeable without very close inspection. The taper on the short cavity was increased to match the other cavity, and what a change! The shorter cavity now filled well ahead of the other cavity.

With the D-style of sub-gate the taper is taken out of the equation for flow restrictions because there is not a gradual taper of mass like with a standard sub-gate.

If you have cavity pressure transducers in multiple cavities it is possible to better understand the impact of the complete process effects on imbalance and plastic conditions. The goal is to see consistent cavity fill times, rise in pressure, peak pressures, and cavity pressure integrals.

12.8.1 Family Molds

Family molds are molds that have cavities that are different in size, shape, and volume. Often a customer will prefer to bundle cavities in a mold to save tooling costs. The practice of bundling cavities in a family mold will result in cavity imbalance. If cavities have different volumes the fill time between them will be different, which will result in different plastic conditions in each of the cavities.

A family mold will always be unbalanced even when using flow analysis software to try to create a uniform fill to all cavities. Even when fill time is the same for all cavities the pack rate, gate seal time, and cooling can still all vary between the cavities.

The best solution is to avoid running family molds. If family molds cannot be avoided, understand that the process window will suffer. Expect problems with dimensions between cavities as well as situations where some cavities short out and others flash. Family molds will always be a compromise between cost and quality, and in the long run the cost of quality may offset the initial savings in mold cost.

References

[1] Beaumont J. "*Runner and Gating Design Handbook*", 2nd Ed., (2007) Hanser, Munich.

[2] Beaumont Technologies Inc. 1524 E.10th st. Erie, PA 16511 Tel (814)899-6390 *http://www.beaumontinc.com*.

[3] RJG Inc. Master Molder Manual.

13 Cavity Instrumentation

Cavity instrumentation is one significant tool available for injection molding and normally comprises the use of either cavity pressure transducers or thermocouples. This chapter will provide a basic understanding of cavity instrumentation. Many examples of data from in-mold instrumentation are used throughout this book.

For a complete understanding of in-cavity instrumentation consider taking a course specializing in scientific molding and molding instrumentation.

13.1 Cavity Pressure Technology

During the 1960s Rod Groleau conducted a number of studies on cavity pressure and published a paper called "A Report on Injection Molded Part Size Control". In 1969, Groleau co-founded a company called Control Process Inc., and began producing the industry's first strain gauge cavity pressure transducer and monitoring system. In 1989, Groleau founded RJG Technologies and began supplying cavity pressure transducers and monitoring equipment to the injection molding industry [1].

Cavity pressure transducers have created a revolution in the molding industry. With an instrumented mold it is now possible to understand what is occurring in the mold. Without in-cavity instrumentation there is a gap between the machine settings and the final part results. Cavity pressure transducers help fill this gap. What occurs in the mold has always been a closed "black box" but now with transducers it is possible to see inside the box.

Cavity pressure monitoring capabilities have come a long way over the years. Not too long ago the use of paper strip chart recorders for capturing process data was state of the art. Progression over the years led to computer-based monitoring systems that at first required careful calibration of transducers and intimate

knowledge of the equipment to set up correctly. Current monitoring equipment utilizes auto calibration and plug and play smart transducers making cavity pressure monitoring easily accessible.

See Figure 13.1 for a typical cavity pressure transducer. Be aware that transducers come in different load ratings, different cable lengths, and different styles including strain gauge and piezoelectric.

Figure 13.1 RJG button style cavity pressure transducer

Cavity pressure transducers are most often installed under an ejector pin and the pin transfers the pressure from the cavity to the transducer. Another technique is to install a pressure transducer at the mold surface. In both cases the transducer will provide pressure data of the plastic in the mold. Long unsupported pins can lead to deflection that will show as an uptick in-cavity pressure after second stage is complete (this can also indicate mold deflection).

A web search will show that there is a wide variety of producers of cavity pressure transducers including RJG Inc., Priamus System Technologies, and Kistler Group among others. Decision of the brand, type, and size must be determined before the mold can be machined for installation.

Some of the information that cavity pressure monitoring can provide from inside the mold include:

- Cavity pressure at all points of the cycle
- Actual mold fill time

- Plastic fill rate
- Cavity balance
- Viscosity shift in material
- Gate seal
- Pack rate
- Pressure drop across the cavity
- Cooling rate
- Mold or clamp deflection
- Valve gate actuation
- Process changes
- Machine performance

All of the above data is collected on every shot and can be set up with alarm bands to determine if the part is good or bad while it is still in the mold.

Figure 13.2 shows typical data from an RJG eDART® monitoring system. Curves for machine pressure, screw displacement, and cavity pressure are displayed.

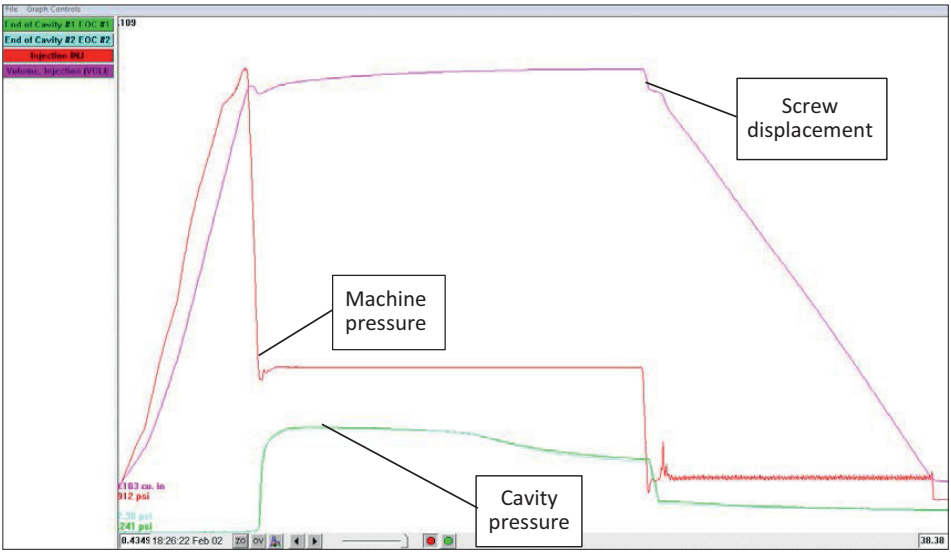


Figure 13.2 Typical process monitoring data from RJG eDART®

Standard points of interest in reading these curves are:

- Data reads from left to right. Start of injection is in the lower left-hand corner (0,0 corner).
- Time increases from left to right along the horizontal axis.

- Displacement along the vertical axis is based on the type of transducer. These values may be pressure, linear displacement, temperature, or any other data being collected. Again the lower left is the zero point for this data.
- Color code in upper left-hand corner details what the data curve colors correspond to.
- Machine pressure will climb from zero to the transfer pressure; second-stage pressure should appear as the flat horizontal line shown in Figure 13.2 for the duration of set second-stage time.
- Screw recovery is represented by the lower horizontal line on the machine pressure curve that is on the lower right of the screen.
- Notice how the machine pressure and screw displacement start to increase at the start of the cycle but the cavity pressure only starts to rise at the transfer position. This indicates that this cavity pressure transducer is located at the end of fill; if it was closer to the gate there would have been a pressure rise during first stage.

Cavity pressure transducers are often located either just inside the gate (post-gate transducer) or as close to the end of fill (end-of-fill transducer) as possible. For process monitoring the end-of-fill transducer will provide data on what occurred through the process and will give accurate information on whether the mold filled or not. Process control usually relies on a post-gate transducer to signal when to transfer from pack to hold. With both post-gate and end-of-fill transducers an accurate picture of the plastic pressure across the cavity is provided. Below is some key information that transducers can provide:

- Post-gate transducers will show a distinct hitch that indicates the change between dynamic filling pressure and static hold pressure (see Figure 13.3). The two cavity pressure curves with higher pressure are post-gate transducers (see labels). Notice how they start to rise early in the cycle and change slope at transfer. This change in slope indicates where the cavity is full and building pressure to the cavity pressure peak.
- Post-gate transducers provide useful data on gate seal. When a gate is sealed the plastic that is packed against the transducer will be maintained after the machine completes second-stage time. If the gate is not sealed plastic will discharge through the gate resulting in a rapid reduction in cavity pressure when second-stage pressure is removed. In Figure 13.3 the post-gate curves drop immediately when second-stage time ends. This indicates that the gates were not sealed on this mold. Determining gate seal time with a post-gate transducer is simply a matter of adding second-stage time until the post-gate cavity pressure maintains a consistent slope. This is a quicker method than the typical gate seal study conducted by weighing parts molded with different second-stage times.

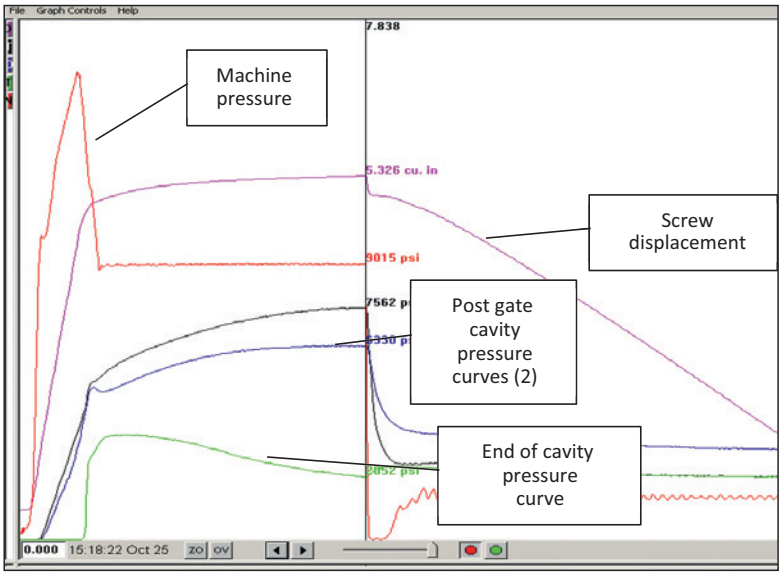


Figure 13.3 Cavity pressure results with post-gate and end-of-fill transducers

- Post-gate transducers provide accurate measures of pressure drop through the runner system and gate. The difference between the machine pressure and the post-gate cavity pressure provides all pressure loss to this point of the flow path.
- End-of-fill transducers provide a measure of the total pressure drop through the entire flow path. See Figure 13.4 for pressure from machine to end of fill.
- Time between the initial rise in post-gate pressure and the initial rise in end-of-fill pressure provide an accurate picture of the actual cavity fill time. This accounts for the total true time that the plastic is actually flowing through the mold.
- The point at which cavity pressure peaks shows the actual time that plastic was being packed at a given location. Notice in Figure 13.3 that the post-gate transducer values continue to rise throughout second stage, whereas the end-of-fill transducer peaks just after transfer. This is an indicator of differential cavity pack pressure that can lead to warp. This particular mold had two hot tips that stayed open throughout the molding cycle.
- The slope of the cavity pressure rise provides an understanding of the pack rate in the mold. The slower the rise in pressure the slower the pack rate.
- The pressure slope after cavity pressure peaks and begins to decrease is a measure of the cooling rate of the plastic. The plastic is cooling and shrinking away from the mold, which is the reason that the pressure is dropping. Faster decreases in pressure mean faster cooling rates.

- Transducers in multicavity molds can provide understanding of cavity balance. All of the above data will help understand the total process balance in each cavity, and not just the fill balance determined in typical cavity balance studies.

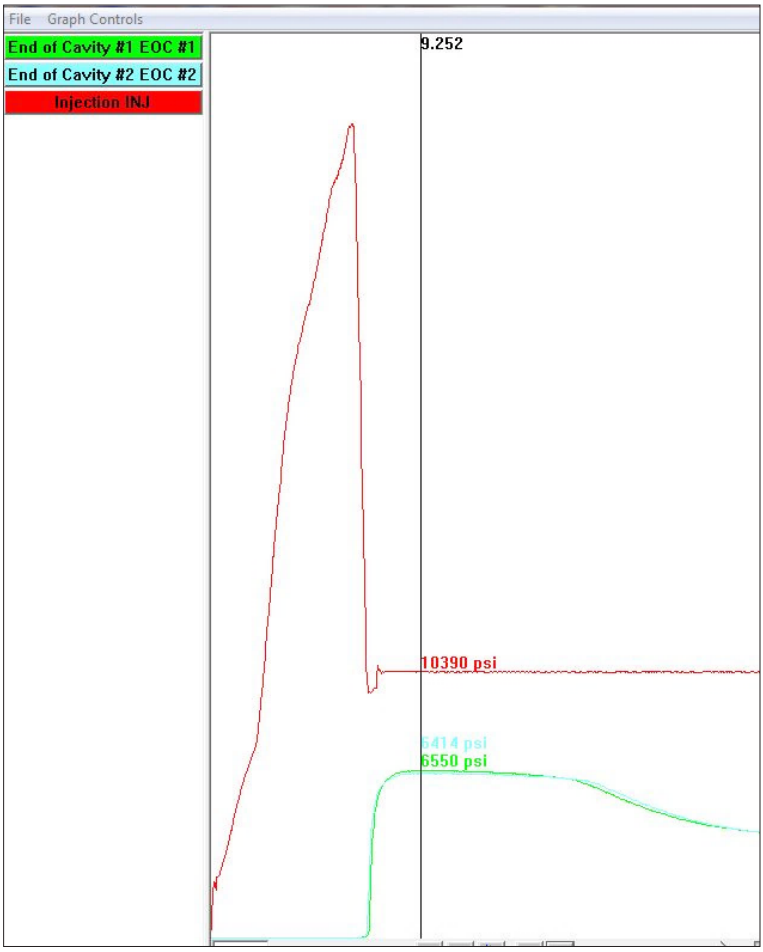


Figure 13.4 Pressure drop between machine and end of cavity

In Figure 13.4 the pressure drop between the machine pressure (10,390 psi) and two end-of-fill transducers (6,550 and 6,414 psi) is easily determined. This data would not be attainable without having installed cavity pressure transducers in the mold. Another benefit in this case is to see that both ends of the cavity are being filled, packed, and cooled exactly the same. Using this data, it is easy to determine that there is approximately 4,000 psi in pressure drop from the machine to the end of the cavity.

Process problems are easy to detect with process monitoring equipment. Figure 13.5 shows a process where the screw run time increased due to a poor material pickup. The next shot was back to normal other than a change in end-of-cavity pressure. Alarm bands can be established to automatically reject parts that fall outside the established process window. Note also in Figure 13.5 that there are three cavity pressure transducers: two near the end of cavity and one just inside the gate.

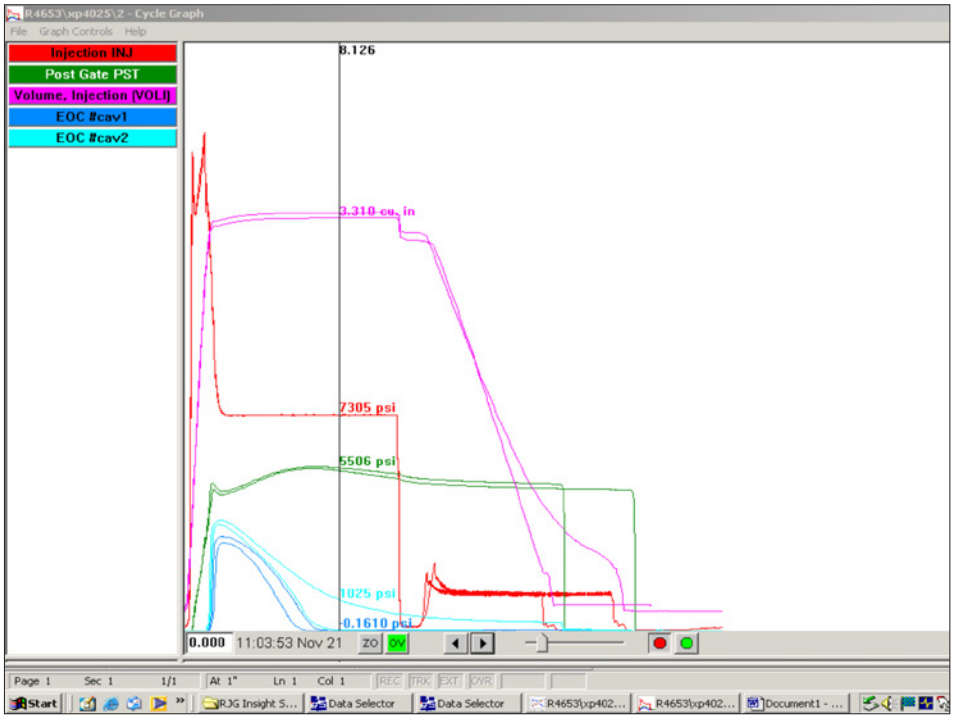


Figure 13.5 Process monitoring showing process change

Process changes are easily detected with process monitoring. In Figure 13.6 a velocity change was made and is visible on all of the curves. This change would be easy to detect on the machine as well but with process monitoring it is possible to detect the impact on the cavity pressure as well as having clear documentation of exactly when the change occurred (helps reduce the "I don't know who made that change" excuse). This is another situation where alarm settings could be used to automatically reject the parts produced with the alternate process.

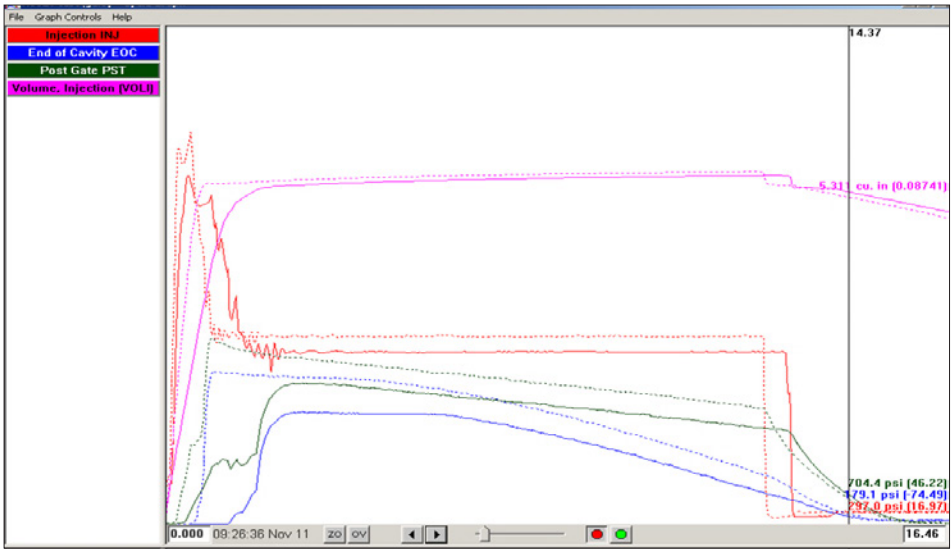


Figure 13.6 Process monitoring data showing change in fill velocity

■ 13.2 Cavity Thermocouples

Thermocouples are a second means of instrumentation within the cavity, which give temperature data from the mold and are normally located at the mold surface in a non-class "A" area. They are set up in contact style on the mold surface and will leave a witness mark, but this is not usually a problem on the ejector side of the part. Figure 13.7 shows an example of an in-mold 1.5-mm diameter thermocouple.

Figure 13.7 1.5-mm in-mold thermocouple

The data obtained from cavity thermocouples are not as detailed as is the case with cavity pressure; however, thermocouples will provide the following:

- Actual mold steel temperature
- Actual fill time to the thermocouple location
- Change to melt temperature or cooling rate

An advantage of cavity thermocouples is that they are extremely fast-acting, which makes them very useful for setting up valve gate actuation. By using cavity thermocouples positioned throughout the mold, sequential valve gate actuation can be very accurate, and the gate does not actuate until the plastic has reached a specific point in the cavity. The equipment can be set up to delay firing based on time after a temperature rise at a given thermocouple. This is a very repeatable way of setting up sequential valve gate actuation.

In Figure 13.8 process data for a variety of sensors is shown including cavity temperature, cavity pressure, machine pressure, and screw movement. Notice that the mold temperature curve stays horizontal until the plastic hits it. The temperature rises quickly once the plastic contacts the thermocouple. In this example the thermocouple appears to be very near the end of the cavity because the temperature seems to rise just after transfer. This fact would allow detection of short shots easily, because if there was no temperature rise at the end-of-cavity thermocouple there was no plastic present, and therefore a short shot occurred.

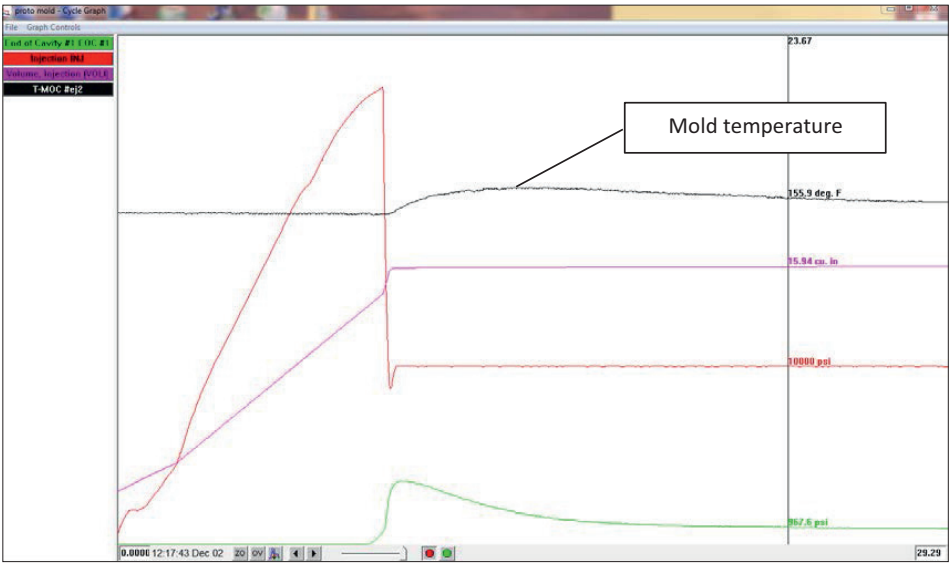


Figure 13.8 Process monitoring data showing cavity temperature, cavity pressure, machine pressure, and screw movement

An unusual application of thermocouples is represented in Figure 13.9. In this particular case auxiliary thermocouples were installed in a hot runner manifold to measure the temperature variation across the manifold. The first thing to note on this data is that even though all of the hot runner zones were set at 510 °F the actual temperatures at the shot the cursor is highlighting varied from 483 to 522 °F. Also notice that each of the curves has a large amount of variation over time. In this particular case there was an ongoing issue with burning and splay types of defects in one area of the part. After the thermocouples had been installed large swings in temperature were detected over time with specific zones changing over 200 °F while the mold was running. Observation of this variation showed no correlation to readings on the hot runner controller, but 100% correlation with the burning and splay scrap. The hot runner controller was replaced with another unit which resolved the issue. Without utilizing the capabilities of the process monitoring equipment this problem would not have been resolved because there was no reason to suspect the hot runner controller.

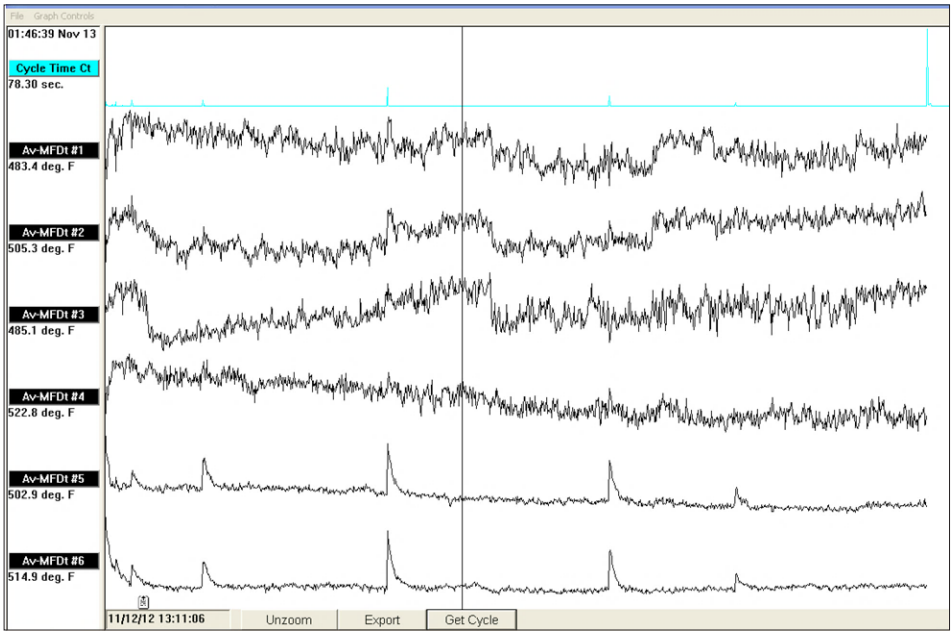


Figure 13.9 Six auxiliary thermocouples installed on hot runner manifold

Another valuable piece of data that can be obtained from in-mold thermocouples is actual mold temperature. Without thermocouples in the mold the best way to check steel temperature is to stop the machine after running on cycle and use a surface pyrometer to check the steel temperature, but as soon as the machine is stopped the mold is cooling from its true running temperature. With a thermocouple

installed in the mold it is possible to always detect the true steel temperature. This also allows for the determination of the number or shots required for full process stabilization (see Figure 13.10 for temperature rise over 20 shots).

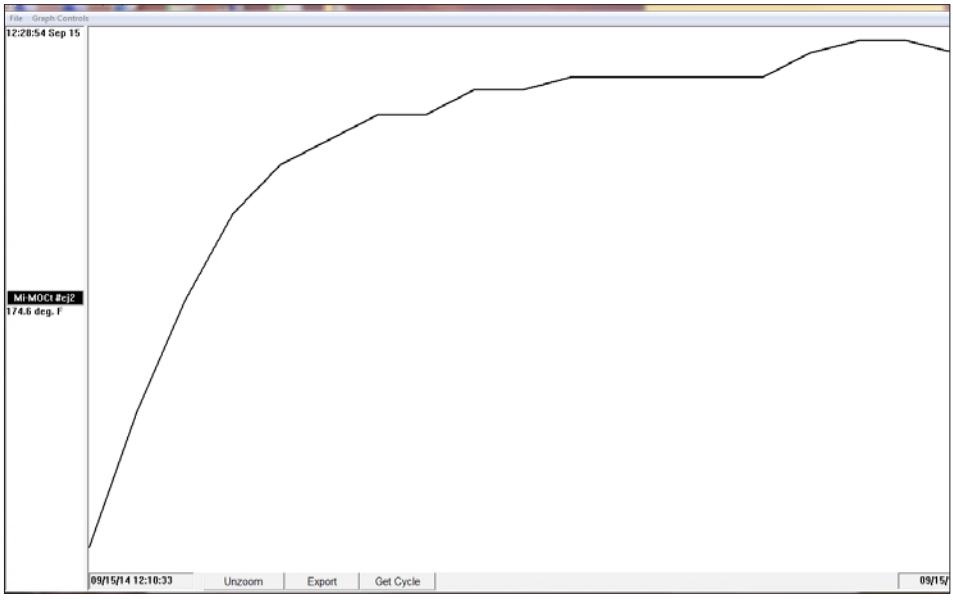


Figure 13.10 Thermocouple data for 20 consecutive shots

13.3 Process Documentation

One major benefit of utilizing a process monitoring system is the ability to document all of the process data for every shot and retain that data forever. This data can prove beneficial for review of any field failures or otherwise defective product. For highest levels of data compliance, a label with specific shot reference information can be printed and affixed to the molded part.

Some molding shops will require process technicians to document hundreds of process settings from the machine and secondary equipment settings. Using in-mold data can provide a more insightful collection of process data and simplify the process of collecting the data. The mentality becomes more of checking that the automatic process alarms and rejection are turned on and working.

The question to ask is: is it better to manually document certain conditions once per shift or allow an automated process monitoring system validate every shot?

13.4 Process Control

In-mold instrumentation can be used for process control. Both cavity pressure transducers and cavity thermocouples can be used for process control. The four typical means of process control are:

1. Decoupled III® processing
2. Valve gate control
3. Gas assist control
4. Mold temperature control unit (TCU)

In Decoupled III® processes the mold is filled with velocity control and still transfers by position. There is a controlled velocity second-stage pack step that packs the part until the cavity pressure reaches a set transfer pressure. After reaching the cavity transfer pressure the machine will be transferred to a pressure-controlled hold stage. The hold stage lasts for the established hold time usually until gate seal. With this method of processing the packing of the part is controlled at a given rate which provides a very controlled process, and the mold will always be packed to the same pressure. An output signal from the monitoring equipment to the machine will allow for the transfer to occur based on the cavity data. Decoupled III® processing is a methodology that RJG Technologies specializes in providing training and equipment.

For controlling valve gates the process monitoring system will use output signals to fire valve gate controls. The process monitoring equipment can be set up to fire the valve gate controls based on when the plastic reaches the transducer or thermocouple. This allows firing of valve gate controls in a very repeatable manner based on what the plastic is doing in-mold rather than on screw position as is normally done. When running a sequential valve gate sequence, the ability to control firing based on plastic conditions will help to eliminate concerns with splay, flow lines, and other process defects and inconsistencies.

With in-mold instrumentation overflows for gas assist can be actuated based on plastic position in the mold. A thermocouple or cavity pressure transducer can provide the direct feedback from inside the mold cavity to provide more accurate timing for overflow actuation. Using an output device tied to the process monitoring system the overflow can be actuated allowing for consistent processing.

In-cavity thermocouples (or thermistors) can be used to provide actual mold temperature data to a mold temperature control unit (TCU). Most TCUs are set up to control based on the output water temperature, however, for more accurate data the mold temperature can be the control point on some TCUs. Using actual mold temperature will provide for a more consistent control of the process over time by reacting to changes that occur downstream of the TCU including:

- Changes in flow rate
- Scale buildup in water lines
- Changes to heat input from higher melt temperatures or faster cycle times

13.5 Additional Monitoring Options

With the ability to take any 0–10-V signal into process monitoring equipment there are many opportunities to monitor details in the molding process or environment. Some options to consider include:

- Dryer monitoring. Use an analog input device to take the temperature and dew point of a dryer into process monitoring equipment. This can be a great way to test dryer performance.
- Melt temperature. An in-stream melt temperature device is an ideal piece of process monitoring. Installation of an in-nozzle thermocouple that is within the melt stream allows for measurement of actual melt temperature during the process. Unfortunately, these thermocouples tend to not last long in the harsh environment of the melt stream.
- Water flow through the mold. Water flow is one of the key items to cooling rate of the plastic. Monitoring the flow allows ongoing detection of any concerns relative to this process parameter. Figure 13.11 shows an example of a typical digital flow meter, and when purchased with output capabilities this unit can be connected to process monitoring equipment.

Figure 13.11 Digital flow meter

- Numerous other potential process inputs from secondary devices can be fed back into the process monitoring equipment. Anything that provides a 0–10-V output signal can be monitored with a 0–10-V input module. Anything with linear motion can be monitored with a linear stroke transducer.

References

[1] https://www.rjginc.com

14 Mold Cooling

14.1 Mold Cooling Importance

At its core injection molding is a thermodynamic process that relies on heat exchange. One of the critical aspects of a mold is to act as a heat exchanger to remove the heat energy from the molten plastic. If a mold is not doing an efficient job acting as a heat exchanger the cycle time of the molding process will be limited.

It is important to understand that a part can only be cooled as fast as the plastic can give up heat energy and as fast as the mold can transfer heat energy from the plastic.

The mold must be cooled as uniformly as is possible. If the mold has hot spots these locations will end up dictating the cycle time due to the reduced cooling capacity.

In most molding processes the overall cycle time is at least 50% cooling time and often up to 80%; see Figure 14.1 (bear in mind that the plastic is cooling as soon as it contacts the mold surface). This means that there is a great opportunity available to optimize cycle times if cooling efficiency is optimized.

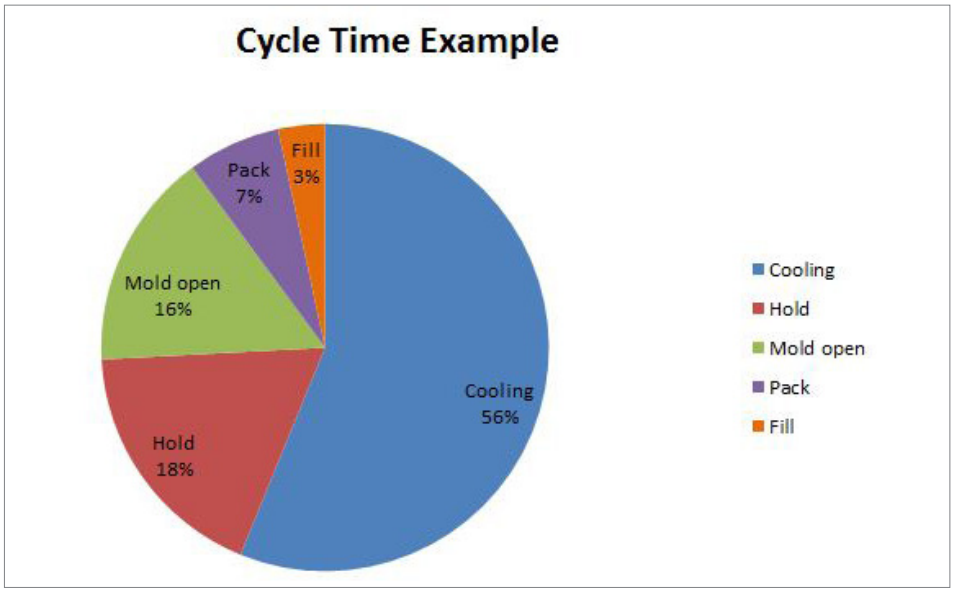


Figure 14.1 Example of cycle time breakdown

14.2 Water Flow

One of the key factors for optimizing cooling is to ensure turbulent flow within the water lines. Turbulent flow is more capable of transferring heat than is laminar water flow. It is vital to ensure that the temperature control unit (TCU) is adequately sized to provide abundant water flow through the mold. If water flow is lacking the resulting laminar flow will not transfer heat as well as turbulent flow.

Caution should be used when exchanging TCUs on a molding machine. Many molding shops will have a variety of TCUs, often with different maximum capacities. If a 7-hp TCU is replaced with a 3-hp TCU the mold is not going to be provided with the same level of flow. If the mold requires the 7-hp TCU to provide adequate flow swapping to the smaller unit may lead to a variety of molding defects. For this reason, it is important to clearly label all TCUs in the plant to ensure if a unit is swapped it is swapped with the correct size.

The way to determine if water flow is turbulent is to calculate the Reynolds number (Re). Equation 14.1 is the formula for calculating the Reynolds number for any given flow condition. There are useful charts available that provide a quick reference to the minimum flow required based on water line size and coolant temperature; the processors who are analyzing cooling should be provided with these to

save them from manually calculating the results. Higher flow rates improve cooling capability but there is a point of diminishing returns where the extra heat removed is not worth the energy to generate the higher flow. One interesting thing to note on this equation is that cooler water temperatures require higher flow rates to achieve turbulence. Another important thing to consider is that adding antifreeze to the water will require nearly twice the water flow to ensure turbulent flow. There are cases where people are trying to reduce the water temperature and end up losing turbulent flow, which in turn restricts the cooling capability of the mold.

$$\mathrm{Re} = \left(3160 \times \mathrm{Q}\right) / \left(\mathrm{D} \times \mathrm{n}\right) \tag{14.1}$$

where:

Re = Reynolds number

Q = flow rate (GPM)

D = diameter of water line

n = kinematic viscosity of cooling medium

Re should be higher than 5000 to achieve turbulent flow.

14.3 Documentation

To provide a repeatable cooling process in injection molding it is important to document the following items:

- Water line layout. The water lines must always be connected in the same manner for every mold run, or otherwise the cooling capabilities of the mold may be impacted. Water line layouts must be clearly documented with in/outs, as well as location and number of jumper loops. Hard plumbing all of the jumper loops on a mold is a good way to provide consistent water hookup.
- Water flow. The return flow should be documented for all cooling circuits on the mold. Note that flow should be checked with all circuits open, not just the circuit being evaluated. In some cases, people will check flow through an isolated cooling line without opening the rest of the lines being fed from the TCU in question, and the result is a much higher value because all water flow is diverted through the one open line.
- Water temperature in and out. The water temperature should be checked at the inlet of each circuit as well as the outlet of each circuit. The industry standard as taught by experts including RJG Inc and John Bozzelli is a maximum of 4 °F temperature rise between the incoming and outgoing water. If the temperature rise is higher than this the circuit should be reduced in length by breaking the circuit into multiple circuits.

- Mold steel temperature. The actual surface temperature of the mold steel should be checked after running the mold on cycle. The cycle should be stopped and the mold temperatures quickly checked with a surface pyrometer. On a sketch or picture of the mold make a note of the locations where the temperature was checked. Avoid scratching class "A" surfaces with the contact probe, because it is usually a better idea to check just outside the cavity rather than risking damage of the cavity surface.
- Part ejection temperature. Check the part temperature after it has been ejected from the mold. This can be done with a surface probe, an IR temperature gun, or even better an IR thermal imaging camera. The part ejection temperature will provide a roadmap for where the hot spots are on the part. If the hot spots have changed over time it may indicate reduced cooling capacity. In some cases, the thermal checks will indicate very clearly that a water line is plugged leading to a line of decreased cooling across the mold.
- Size of TCU. By documenting the size of the TCU used on the mold it will be easy to verify that the correct unit is being used during future troubleshooting.

14.4 Tooling Techniques

Mold materials can have a large impact on cooling ability. Steel or alloy materials can be considered relative to cooling ability and wear resistance. Copper-based alloys can transfer heat much faster than steels but must have a protective coating when using glass-filled materials. Also, hardened steels are less conductive than steels such as P-20. Hardened H-13 is 15–20% less conductive than P-20 so the material must be a consideration when designing a mold. The material being molded often drives the steel choice due to wear resistance but cooling should also be factored in.

When it comes to cores and inserts the material is also important. Remember if you are able to get a cooling channel into the detail alloys are often not necessary. Bubblers are a very effective component to get cooling into tight areas. Bubblers are an often-overlooked cooling solution when space is restricted. There is also a time and place for ISO pipes but they are not as effective as getting flowing water into an area. Keep in mind with small bubblers that plugging can become a problem if the plant water is not being properly conditioned. If plugging is becoming an issue it is important to understand that all of the mold is being affected by the buildup of mineral deposits and the cycle time of the mold may be compromised. When water issues occur dig into the root cause of the issue and deal with the plant's water treatment.

With bubblers an overlooked area is the difference in volume between the inner diameter of the tube and the outer diameter of the tube to the water line diameter. Use of a high-flow bubbler tube in place of traditional brass bubbler tubes can allow for better water flow. One aspect of bubblers that is commonly overlooked is the cross-sectional area of the inner diameter of the tube and the area between the outer diameter and the water-line diameter. This is one reason we always recommend using high-flow stainless-steel bubbler tubes versus the common brass bubbler tubes. For example, with a 0.125-in diameter tube, the high-flow tube has an inner diameter of 0.109 in, whereas a brass tube at best has an inner diameter of 0.069 in. The high-flow tube thus has 250% more cross-sectional area, which translates to an increase of 250% in GPM. That is why we never use standard brass tubes: they decrease your GPM potential.

You must also consider the cross-sectional area between the tube and the water-line diameter so that you are using the proper tube size. Some will assume that with a 0.250-in hole you would use a 0.125-in bubbler tube, thinking it is half of the diameter. But you must consider the area instead.

There have been many cases where creative water line design led to major improvements in the cooling capability of a mold. In real-world applications water lines of as small as 0.060 in have been used to successfully cool difficult areas. In a case study cycle time was reduced on a mold from 40 seconds to 17 seconds by adding 0.060 in water lines. Do not overlook options that maybe outside the box. For example, Figure 14.2 shows a thin bubbler inserted into a skinny core. Note that bubbler tubes are as small as 0.005 in outer diameter with a 0.002 in inner diameter!

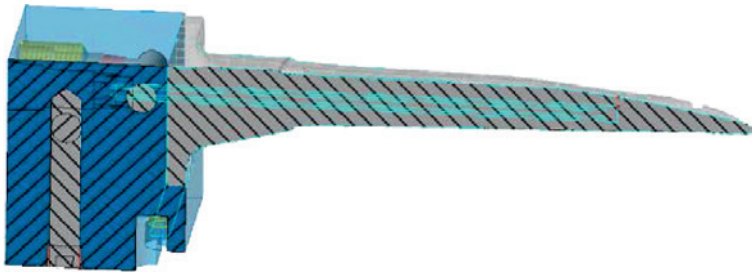


Figure 14.2 Small cooling line in thin core

14.5 Other Cooling Concerns

A major factor on the cooling ability of the mold is ensuring that the cooling lines do not have a mineral scale buildup within the lines. If scale builds up within cooling lines it will act as an insulator and reduce the cooling capacity of a mold. It is vital that water is treated to reduce the opportunity for scale buildup; otherwise diminishing cooling capability will occur over time as the mold runs. Scale buildup can be removed with an acid flush system that will circulate acid through the cooling lines to eliminate the built-up minerals. Figure 14.3 shows how a water line can become choked off by scale buildup.

Figure 14.3 Severe scale buildup in water line extension

Ensure that the plant water is being conditioned with an effective water treatment. Hard water typically contains calcium and magnesium both of which can plug water lines and build up as scale within the water lines leading to a reduction in cooling capability.

15 Black or Brown Specks

15.1 Description

Black or brown specks are spots of degraded plastic or dark contamination that show on a molded part. This issue will be much more visible on white or clear molded parts.

Also known as: carbon buildup

Mistaken identity: material contamination

Figure 15.1 Black specks

15.2 Black Specks Troubleshooting Chart

Table 15.1 shows the black specks troubleshooting chart.

Table 15.1 Black specks troubleshooting chart

Molding Process	Mold	Machine	Material
improper machine shutdown	dust from wear surfaces	hang up areas, screw, barrel, etc.	contamination
high melt temperature	hot runner temperatures	barrel heater control	wrong material
long residence time	hot runner hang up areas		regrind fines

15.3 Black Specks Troubleshooting

Black specks are typically a sign that the material has at some point been severely degraded. Most troubleshooting efforts will focus on time at temperature for the material and the details that affect both temperature and residence time.

15.3.1 Black Specks Troubleshooting Molding Process Issues

Black or brown specks usually are a sign that the plastic has been degraded to a charred state. This is most often a case of material that has been at temperature for too long of a time. There are several factors in the molding process that can lead to the formation of black or brown specks including:

- Improper machine shutdown
- High melt temperature
- Long residence time

15.3.1.1 Molding Process: Improper Machine Shutdown

For most materials it is critical to properly purge the molding machine at shutdown. If a material is held for an extended period of time at melt temperature, degradation will occur. Not that some materials are much more sensitive to this problem than others; materials such as polypropylene are very stable whereas a PVC will easily degrade and produce charred material. When shutting down a molding machine it is important to purge the machine clean of the material and follow up with an appropriate purging compound (could be a commercial com-

pound or something stable like polypropylene). It is vital to establish a documented machine shutdown process for all materials.

In cases where clear polycarbonate (PC) is being molded an effective shutdown procedure is to bank the heats to a lower set point rather than completely turning them off. By banking the heats with clear PC the residence time that occurs during cool down and warm up is eliminated.

Case Study: Clear Polycarbonate Shutdown

In this particular case the part being molded was a clear PC dirt bin in a 700-ton molding machine. Over the history of running this part black specks were a major scrap problem. After trialing numerous process changes it was decided to bank the heaters on the molding machine to 300 °F whenever the job was shut down. After making this change in shutdown procedure black specks were no longer an issue.

15.3.1.2 Molding Process: High Melt Temperature

While high melt temperature is not typically the root cause of black specks it can be a contributor to the issue. If plastic is exposed to higher than recommended process temperatures it is more likely to degrade. Black specks are material that has been severely degraded usually through an extended exposure to increased temperature. If the material is exposed to excessive temperature the likelihood that it will degrade to the point of producing black specks is increased. Again when dealing with black specks the issue usually goes back to length of time at temperature, but the higher the temperature the less time before the material will degrade.

15.3.1.3 Molding Process: Long Residence Time

Excess time at temperature is what typically creates black specks. Normally, to reach the point where material turns to black specks, it takes more time than the residence time that the plastic sees moving through the barrel. However, keep in mind that black specks are typically a time and temperature problem, so it is important to evaluate what the material is experiencing from a heat history viewpoint.

15.3.2 Black Specks Troubleshooting Mold Issues

Some of the potential areas related to the mold can include:

- Debris from mold wear or shutoff surfaces
- Hot runner temperatures
- Hot runner hang up areas

15.3.2.1 Mold: Debris from Mold Wear or Shutoff Surfaces

Sometimes black specks do not arise from material degradation. An often-overlooked root cause of black specks on plastic parts is dust from the mold wear surfaces. Many times, the miter locks, wear plates, or shutoff surfaces on molds will create a dust of metal that can make its way into the mold cavity where it will be molded into the part. This metal dust will appear as black specks if it is molded into the surface of a light-colored part. Note that when this occurs the location of the problem area is normally above the cavity, which allows the dust to drop into the cavity.

Mold cleanliness is critical to avoid this potential root cause. Wear surfaces should be kept clean and examined for metal "flaking" because this could indicate a larger problem with the mold.

Caution should be exercised whenever offset mold temperatures are used. If the mold halves are being run at differing temperatures thermal expansion will lead to differential expansion that may contribute to excessive wear on shutoffs or lock angles. Especially with larger molds this differential expansion may lead to metal debris from wear dropping into the cavity leading to black specks.

If larger flakes of metal are detected a closer inspection of the mold is warranted. These metal flakes often indicate that something is damaged or wearing at an unusual rate. Detection of abnormal items like metal flaking can help eliminate expensive mold damage. Always look for areas on the mold where galling may be happening. Wear plates made from bronze may be required on certain wear locations on a mold.

15.3.2.2 Mold: Hot Runner Temperatures

Similar to high melt temperature this will usually not be the root cause of the black specks but can be a contributing factor. If the hot runner temperatures are set high or running hot for any reason the material will be pushed closer to the limit where it will degrade. It is important to keep in mind that the time spent in the hot runner manifold is normally much shorter than the time spent in the barrel, but depending on the manifold design the residence time in the manifold can be several shots.

Verify that the mold's hot runner temperatures are set correctly and running to set point. Also check the amp (current) draw on the hot runner controller to determine that the zone is under control. If the current to a given zone is not cycling it may be an indication that the zone is not wired correctly. If a zone is drawing current continuously it may be because the thermocouples for two zones are swapped, resulting in the controller providing constant power to a zone without seeing the temperature rise.

15.3.2.3 Mold: Hot Runner Hang Up Areas

Anywhere in the melt stream where plastic can be trapped can lead to degradation of the material. Any spots in the hot runner manifold that have a mismatch in the steel will potentially lead to material hang ups. Common concerns are corners, especially if they are 90°, mismatch between nozzles and the manifold, and using an undersized nozzle tip orifice relative to the sprue opening.

Many of these hot runner issues should be dealt with upfront during the selection and design of the hot runner. If there are issues with stagnant corners after the manifold has been built, it will be a major hassle to correct.

The correct sized nozzle tip should always be used with a hot runner manifold. If the tip orifice is undersized to the sprue opening there will be a shelf and dead spot at the interface that will trap material potentially leading to black specks. The process needs to be developed with the correct size tip and this size must always be used for subsequent molding runs.

15.3.3 Black Specks Troubleshooting Machine Issues

Potential machine-related causes include:

- Hang up areas on screw, barrel, or non-return valve
- Barrel heater control
- Machine performance

15.3.3.1 Machine: Hang Up Areas

A likely cause of black specks is a spot in the barrel, screw, end cap, or nozzle that is allowing material to hang up and sit for extended periods of time. The two most common causes of hang up areas are damage and mismatch.

A common area for hang ups to occur is when there is a mismatch between the barrel, end cap, nozzle adapter, or nozzle (see Figure 15.2). It is critical to ensure that all of the end components of the machine have uniformly matched surfaces. If there are any shelves where the components are not matched material will hang up and degrade. Material that is trapped in these areas will degrade over time and likely reach a point where there is a heavy buildup of carbon, and this carbon buildup will tend to break loose over time and be carried along by the melt stream producing parts with black specks (see Figure 15.3) for an example of mild buildup on a screw tip). Always utilize components that have the same internal diameters to eliminate the potential for hang up areas.

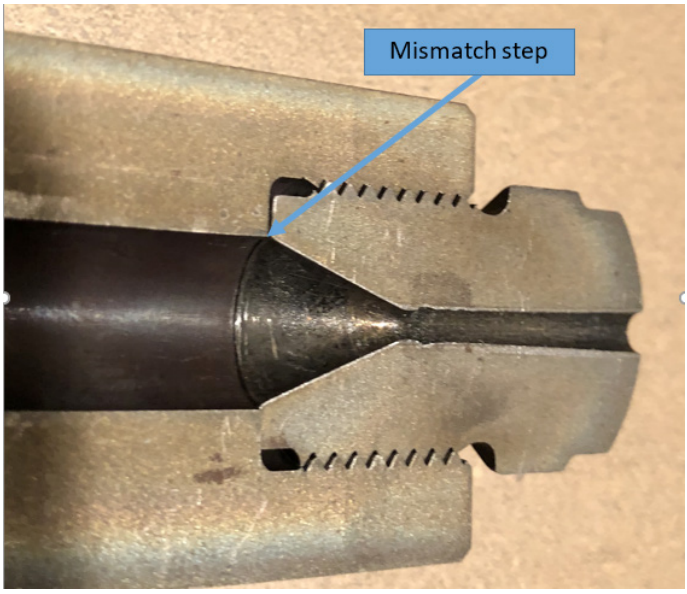


Figure 15.2 Nozzle to nozzle tip mismatch step

Figure 15.3 Non return valve with light buildup

Damage on any of the screw and barrel components can create locations that allow material to hang up and degrade. Similar to mismatch, these damaged areas can trap material for extended periods of time, which in turn leads to the risk of degradation and black specks. Worn screws can also create undercuts that will trap material and lead to black specks. Verification of all components of the melt delivery

system may be required when experiencing ongoing black speck problems. Check for mismatch, excess wear, and damage because often times if any of these are the root cause there will be a buildup of blackened material trapped in these areas. Replacement of the component is the likely solution to the problem.

15.3.3.2 Machine: Barrel Heater Control

A machine must be in control of the barrel heaters to ensure that there are no locations of overheating in a barrel or nozzle. Verify that there are no hot spots on the barrel or nozzle; a good method to check this is by using an IR thermal imaging camera (see Figure 15.4). Note that black specks are usually a later state of degradation; brown streaks and splay may be an early indicator of problems.

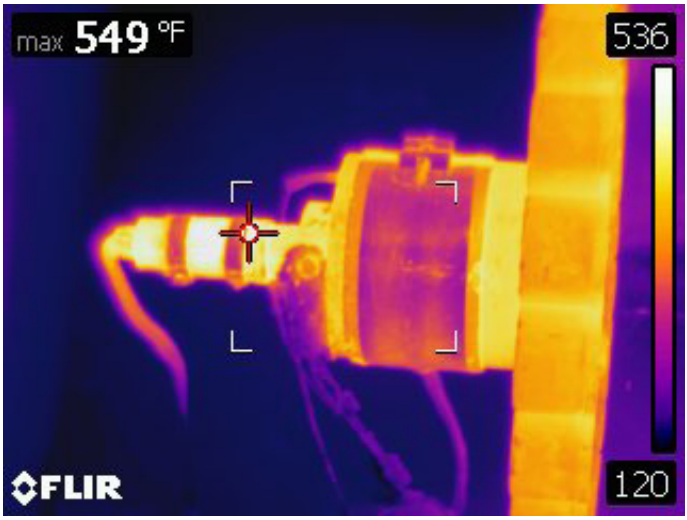


Figure 15.4 Infrared photo of nozzle

15.3.3.3 Machine: Machine Performance

The machine must be running at the established set points. See Chapter 8 on machine performance for more information.

15.3.4 Black Specks Troubleshooting Material Issues

Potential material related concerns are as follows:

- Contamination
- Wrong material
- Regrind fines

15.3.4.1 Material: Contamination

In processing material can be contaminated with a variety of substances including other plastics, dirt, cardboard, etc. These contaminants can create black specks either by simply being a dark contaminant or by burning up at the required melt temperatures. Material contamination can occur in many circumstances including bad regrind, mixed material, improper cleaning of hoppers or loaders, poor plant cleanliness, etc. Many of the issues with material contamination can be fixed by having clear standards and procedures for activities such as cleaning hoppers or always using lids on gaylords (see Figure 15.6). Figure 15.5 shows black contamination in natural regrind.

Figure 15.5 Contaminated regrind

Figure 15.6 Gaylord without lid

15.3.4.2 Material: Wrong Material

If the wrong material is loaded into a molding machine many potential problems can occur including black specks. If the material is overheated it can degrade, which could lead to black specks. It is important to make sure that the correct material is being used in all cases. Normally there will be a multitude of issues that show up if the wrong material has been loaded, but it is always a good thing to verify when troubleshooting.

15.3.4.3 Material: Regrind Fines

When plastic runners and parts are run through a grinder there can be inconsistent grind and a level of fines or dust (see Figure 15.7). These regrind fines will have a tendency to melt early in the barrel and as a result will tend to degrade more easily.

Dusty regrind is often times a sign of dull grinder blades that are beating the material more than cutting it. Maintenance of grinders should include blade sharpening and setting of blade to mesh gaps per recommended specifications.

Figure 15.7 Regrind fines

16 Blush

16.1 Description

Blush is a cosmetic defect commonly found just inside the gate location of a part, resulting in an appearance that may be dull or discolored. Sometimes blush can be found in areas where a wall stock transition occurs. See Figure 16.1 for an example of blush.

Also known as: gate blush, gate shear, halos

Mistaken identities: jetting, flow lines, flow marks

Figure 16.1 Gate blush

16.2 Blush Troubleshooting Chart

Table 16.1 shows the blush troubleshooting chart.

Table 16.1 Blush Troubleshooting Chart

Molding Process	Mold	Machine	Material
injection velocity	gate geometry	nozzle	
nozzle temperature	hot runner tip temperature	machine performance	
mold temperature	cold slug well		
melt temperature	gate location		

16.3 Blush Troubleshooting

Blush is a problem that will typically occur on a part just inside the gate. It presents as a non-uniform fanning out from the gate and appears as a different color or gloss level compared to the plastic surrounding it. Blush can also appear at wall stock changes on a part. Be careful when troubleshooting blush because there are cases when jetting gives a blush-like appearance.

16.3.1 Blush Troubleshooting Molding Process Issues

There are several factors in the molding process that can lead to the formation of blush including:

- Injection velocity
- Nozzle temperature
- Mold temperature
- Melt temperature

16.3.1.1 Molding Process: Injection Velocity

A fill velocity set too high is the most common molding process related cause when dealing with blush.

Before slowing down the fill speed make sure that the gate size and design is adequate, see below under tooling issues (see Section 16.3.2). Processing around a blush issue that has a root cause related to tooling design will create a process that is at risk of other defects, especially short shots. Maintaining high fill velocity is an

advantage both to maintain a more consistent viscosity as well as minimizing cycle time. Avoid processing around tooling issues!

Improving blush whether it is at the gate or downstream from the gate (such as a wall stock change) often will come down to adjustments in velocity settings. Normally slowing the velocity will improve the blush. Sometimes creating a velocity profile to allow a slow fill through the area of concern will be the best solution. Remember that when adding velocity profiles, the level of complexity in the process just increased, which will make it more difficult to repeat.

When making process adjustments for velocity settings it is often advantageous to make a large change to the set point to see if it will have an effect. Often times a change of 50% to the initial velocity will be appropriate to determine impact. There will be cases where the velocity that eliminates the blush will only be a small fraction of what the desired fill velocity is for the mold.

If the blush is appearing at a downstream location such as a wall stock change it may be necessary to profile the velocity through this area of the part. To determine where to set the profile it will be important to run short shots to determine the machine profile settings to allow the velocity to be adjusted at the right point of the shot. The following sequence describes how to find this point:

1. Set pack and hold pressure to zero or the minimum setting for the machine.
2. Adjust the machine's transfer position to a larger set point.
3. Examine the part produced; it should be a short shot that is shorter than the fill only part (95–98%).
4. If the short shot is not at the point where the problem is occurring continue to complete steps 2 and 3 until the short shots reach the point where the problem occurs.
5. The setting for the adjusted transfer position will be the setting that needs to be used for the velocity profile change. Enter this position as the point where the velocity profile slows down.
6. If possible increase the velocity after the area of concern by adding a step in the profile where the velocity is changed back to a faster setting.
7. Reset the original transfer position and check a fill only shot (95–98% full), and keep in mind that the slower velocity may result in a shorter fill only shot. If required, adjust the transfer position to give the desired fill only shot.
8. Put pack and hold pressure back to the set point.
9. Verify the parts are better. It may take some fine tuning of the profile set points as well the velocity through the problem area to resolve the issue.

16.3.1.2 Molding Process: Nozzle Temperature

An incorrect setting on a nozzle temperature can result in gate blush. The first plastic in the mold is normally where blush occurs so this material is the material that would be located in the nozzle between shots. In general, the nozzle temperature should be set at the melt temperature of the material.

Either raising or lowering the nozzle temperature can reduce the severity of blush. Try lowering the nozzle temperature and inspecting the parts, and if the blush is better try running this way to see if the problem is resolved, but if it is worse try raising the nozzle temperature above the original set point.

If the nozzle temperature seems to help but there is variation over time confirm that all controls for the nozzle temperature are being maintained. Watch the machine controller for variations in the nozzle temperature zone, and also look for cases where the nozzle heater zone is calling for heat constantly or is never calling for heat. Examine the nozzle heater band to ensure that it is the correct length for the nozzle, is tightly attached, is the correct wattage, and is working.

16.3.1.3 Molding Process: Mold Temperature

The mold temperature will impact the initial plastic entering the cavity. Both too low and too high mold temperatures can lead to gate blush problems, so it can be important to cover both extremes. Gate blush is a strange phenomenon that can be magnified by the steel temperature and both too low and too high steel temperatures may lead to a poor formation of the initial plastic through the gate.

Verify that the process is at the documented set point. During process development try lowering and raising the mold temperature to determine if it is impacting the blush.

In many cases with materials like ABS, PC/ABS, or TPO warmer mold temperatures will result in a better-looking part surface inside the gate location. In some cases, it appears that colder mold temperatures have more impact on the morphology of these types of materials with rubber impact modifiers.

16.3.1.4 Molding Process: Melt Temperature

Similar to mold temperature the melt temperature can impact the quality of the initial formation of plastic after it leaves the gate.

The melt temperature should be set to the documented process. If the melt temperature is at the correct setting try adjusting the material either up or down. The change in melt temperature may impact the blush of the material entering the cavity. If the blush improves try to run the process for an extended time to ensure that the process adjustment works long term.

Excessive melt temperature may release gas that causes a buildup just inside the gate that leads to blush. Also, high melt temperature may allow erosion of the first material formed inside the gate causing a disruption to the surface.

16.3.2 Blush Troubleshooting Tooling Issues

The main mold cause of blush relates to gate dimensions:

- Gate geometry
- Hot runner tip temperature
- Cold slug well
- Gate location

16.3.2.1 Mold: Gate Geometry

Often the design of a gate will lead to blush problems. If a gate has a long land length it may increase the blush on the part. Normally, gate land length should be held to a maximum of 0.030 in and a minimum of 0.005 in distance from the cavity.

Any setback of the gate from the class "A" mold surface may lead to blush. If the gate is not flush to the mold surface there will be an area of back fill that will leave a disturbance to the plastic entering the part. If possible reduce this setback to the minimum acceptable offset from the class "A" surface as this will normally help with the blush appearance.

Try widening the gate, because a wider gate will allow more material flow and may minimize blush at the gate. Gate thickness may also play into the blush; however, do not assume that a thicker gate will eliminate blush. Gate thickness can go either way when it comes to gate blush, and there are times that thinner gates will improve blush. Figure 16.2 shows typical gate blush.

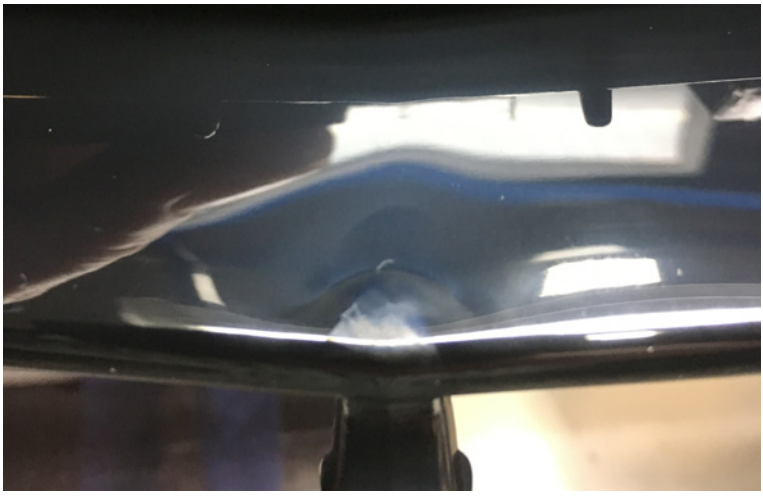


Figure 16.2 Gate blush

Whenever running with a hot runner system be sure to check that the orifice of the hot tip is not causing a restriction that in turn is leading to the problem. People often overlook the hot runner tip on the mold.

In rare cases runner geometry may contribute to gate blush (see case study below).

Case Study: Runner Geometry

This was a case of a polypropylene part where a blush defect was occurring after gate seal during screw recovery. The theory was that the large runner size was allowing the plastic pressure to break through the sealed gate causing the blush appearance. The runner was 0.375 in diameter and approximately 3 in long. The full-round runner was welded up to produce a half-round runner, which helped the blush appearance some without increasing the fill pressure or causing any other problems. A standing rib was welded in the mold to core out the existing half-round runner. When the mold was run again the blush was gone and there was only a slight increase in fill pressure. The smaller runner prevented the plastic pressure during screw recovery from breaking through the sealed gate. This is not a common case of gate blush but demonstrates that sometimes the total picture must be examined via the STOP process.

Another thing to consider trying when working on blush problems is to break the sharp edge of the gate as it enters the cavity. Breaking these gate corners can help reduce the shear rate at the corners of the gate. Reduction of the shear rate may decrease the blush effect as the plastic travels through the gate.

Gate land length is the other area that must be addressed. The land length is the distance where the gate narrows to its final thickness between the cavity and the runner. Long gate land length can also contribute to blush issues. The land length of the gate should be minimized to avoid premature restriction of the melt flow. Long land lengths will also lead to increased pressure drop through the gate. Normal industry standards will recommend gate lands of approximately 0.030 in. Avoid long land lengths to minimize the opportunity for blush.

16.3.2.2 Mold: Hot Runner Tip Temperature

If a hot runner tip and drop have different temperatures a blush-like appearance may occur just inside the gate. Try to minimize differences between actual melt temperature and the set point of the hot runner system.

If running a hot runner system verify that the hot runner temperatures are set correctly. Try adjusting the tip temperature to determine impact on the blush. Again remember that hot runner temperatures should be set equal to the actual melt temperature.

When evaluating hot runner systems verify there is an adequate number of heaters and thermocouples and that they are properly located. Also verify that heaters and thermocouples are properly installed and tightened to ensure accurate results.

Verify that the hot runner zones are all reading correctly. Look for zones that are running without thermocouple control as they will not provide accurate closed loop temperature control. The hot runner zones should be checked for amp draw to establish that they are heating correctly. Examination of the thermocouple location on the drop may need to be conducted to understand where the actual drop temperature is being measured.

16.3.2.3 Mold: Cold Slug Well

A cold slug well is located in a mold at the end of the sprue entering the runner system. The cold slug well is intended to catch any frozen slugs of material from the nozzle tip. If a cold slug well is not available or designed to be as large as the sprue it will not be effective in trapping any cold slugs that are injected from through the sprue. Figure 16.3 shows an example of a cold slug well with a trapped cold slug; in this case the cold slug was actually a hot tip vestige that had stuck at the hot drop. If a slug of material like this should make it to the part and it makes it through the gate there will be flow defects that may appear as blush. Runner systems should have a cold slug well to prevent this material from making its way to the part. Using the STOP method will help avoid processing around a mold that does not have an adequate cold slug well.

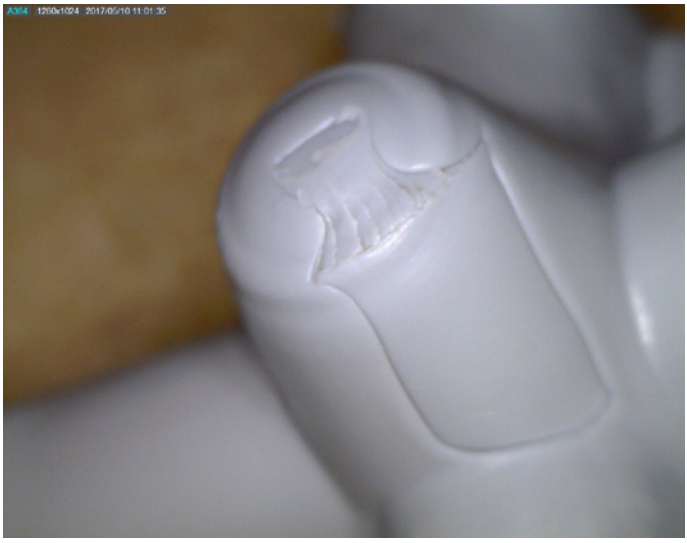


Figure 16.3 Cold slug trapped in cold slug well

16.3.2.4 Mold: Gate Location

When gating directly into a class "A" surface there will be struggles with appearance. One of the most effective ways to avoid problems with gate appearance is to locate gates in an area of the part that is not class "A". To accomplish this, look for opportunities to gate into tabs, ribs, side walls, or other part details to provide a well-established melt front to hit the show surfaces of the part.

Frequently, when a cashew gate is coming up directly behind the class "A" surface of the part rings of blush will be apparent, even looking like a bull's-eye in severe cases. This design will often result in narrow process windows to yield acceptable parts. Locating the gate in a less objectionable area would have resulted in a better process window and less overall problems.

16.3.3 Blush Troubleshooting Machine Issues

Machine contributions to blush may include:

- Nozzle
- Machine performance

16.3.3.1 Machine: Nozzle

Verify that the machine has the correct nozzle setup. The nozzle length, style, tip type, tip orifice, and heaters should all match the documented process. This an area that some processors may consider overkill to document but frequently in injection molding troubleshooting it is the small things that can lead to many hours of time wasted. Thorough documentation will allow for a much easier job for 4M troubleshooting.

Verify that all of the nozzle components are set up to the required standard. Use a nozzle orifice gauge to verify that the tip orifice is correct. Check for complete coverage of nozzle with heater bands. An additional item to check is the nozzle thermocouple; ensure that it is correctly installed within the nozzle. Loose thermocouples can give wrong temperature readings and cause a nozzle to run at the wrong temperature. Verify that the nozzle tip is not the wrong type, specifically a mixing tip or a nylon tip when a full taper is called for.

Watch for temperature variation across the length of the nozzle. There are times when the thermocouple at the rear of the nozzle is not providing an accurate measure of the actual nozzle temperature. Use of a thin immersion thermocouple can allow detection of variation in nozzle temperatures when troubleshooting.

16.3.3.2 Machine: Machine Performance

To avoid blush it is always important to ensure that the machine is achieving the required set points for all parameters.

For details see Chapter 8 on machine performance.

16.3.4 Blush Troubleshooting Material Issues

The biggest concern with materials is that some materials are much more prone to display blush than others. Some of the worst materials for blush seem to be blends such as PC/ABS. There is a school of thought that believes that the blended material experiences a morphology separation. High resolution magnification has shown that there appears to be phase separation and alignment between the polycarbonate and ABS phases that may contribute to the appearance of blush. See Figure 16.4 for furrow-like appearance of blush under magnification.

TPOs are another material type that tends to blush easily. It seems that, between the filler content and the rubber content of TPO, blush will be more common at the gate. Trying a different version of TPO may provide better-looking parts. If a material change becomes necessary close consultation with the customer is required.

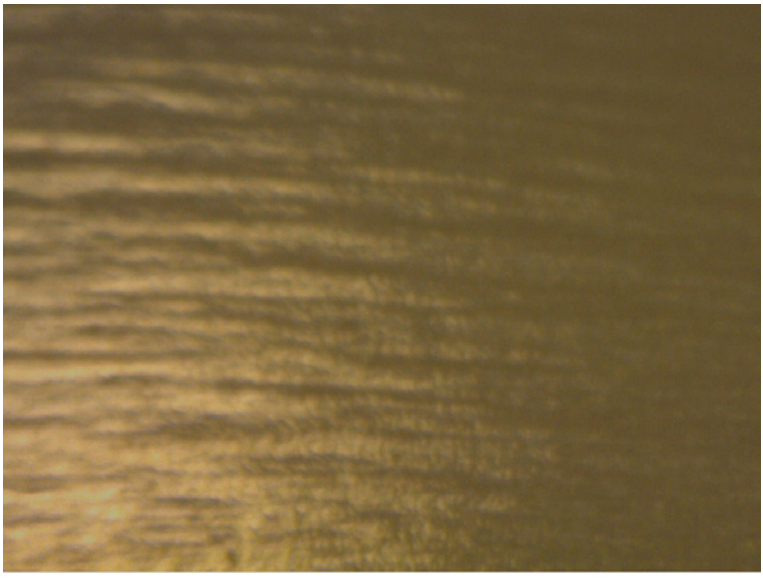


Figure 16.4 Magnified blush showing furrow-like characteristics

17 Brown Streaks

17.1 Description

Brown streaking is a discoloration of the material usually but not always resulting from material having become excessively hot or dead spots where material hangs up. Note that these discoloration streaks may appear as a color other than just brown. See Figure 17.1 for an example of brown streaking.

Also known as: discoloration, streaking, degradation

Mistaken identity: contamination

Figure 17.1 Example of brown streaking just inside of gate

17.2 Brown Streak Troubleshooting Chart

Table 17.1 shows the brown streak troubleshooting chart.

Table 17.1 Brown Streak Troubleshooting Chart

Molding Process	Mold	Machine	Material
barrel temperatures	thermocouple location	heater bands	wrong material
hot runner temperatures	manifold leaks	thermocouples	contamination
residence time	lubricants	machine performance	color concentrate
back pressure	bleeding of mold saver	hang up areas, screw, barrel, etc.	
recovery speed	hot runner insulator gap	screw design	
	sprue bushing seat		

17.3 Brown Streak Troubleshooting

Brown streaks are usually a sign that the material has been exposed to either excessive temperatures or elevated temperatures for extended times. Based on this fact it is critical to review all of the heat sources that the material is exposed to during processing. Also, closely examine any potential areas that material may hang up and experience extended time at temperature.

As with many common defects the location of the brown streak can be an indicator of where the problem is occurring. If the brown streaks occur just inside the gate area of the part it is a sign that the overheating or hang up area is most likely occurring toward the end of the melt control process.

17.3.1 Brown Streak Troubleshooting Molding Process Issues

The main process-related causes of brown streaks are:

- Barrel temperatures
- Hot runner temperatures
- Residence time
- Back pressure
- Recovery speed

17.3.1.1 Molding Process: Barrel Temperatures

The most common process cause of brown streaks or degraded material comes from running barrel temperatures too high. Each plastic material has a maximum upper temperature that it is able to withstand and if it is exposed to temperatures higher than this upper limit degradation will occur. The first thing to check when dealing with brown streaks is that the barrel temperatures are set correctly and match the specification (either setup sheet or manufacturer's recommendation).

When reviewing barrel temperature set points also ensure that the actual temperatures on each zone are holding at the set point; they should be maintained within ±5 °F. A barrel heater zone that is overriding can be a sign of either a controller issue (see Section 8.6.1) or an issue with excessive shear heating causing the heater zones to override the set point.

17.3.1.2 Molding Process: Hot Runner Temperatures

If the mold has a hot runner system it should be treated as an extension of the barrel heating and must be controlled within the requirements for the material. Hot runner temperatures should be verified along with the barrel temperatures to ensure that the temperatures are set correctly and not overriding.

Be aware with hot runner systems that there are multiple potential points of failure beyond just the hot runner manifold. These failure points include:

- The hot runner controller
- Hot runner cables
- Hot runner connector plugs

The hot runner controller can have zones that are failing to maintain temperatures at the required set point.

Case Study: Hot Runner Controller

In this particular case a mold was running with high levels of scrap for brown streaks and splay. Detailed observation of the scrap showed that the scrap tended to occur in bunches. Everything on the mold and hot runner checked out as working correctly but all indications were that the material was being overheated. To gain further understanding of what was occurring within the manifold additional thermocouples were added to determine what was different on the end of the part at which the brown streaking was occurring. These secondary thermocouples were connected to a process-monitoring system to record the manifold temperatures over time. During a period of high scrap, the data on the process-monitoring system was reviewed to determine if anything was changing. The monitoring thermocouples showed that the end of the part with the degraded material was seeing temperature variation of over 200 °F while the scrap was being produced but maintaining temperature when acceptable parts were being produced.

▶

Examination of the hot runner controller during this time period showed that none of the control zones were registering anything out of the ordinary. To determine if the hot runner controller was the problem it was switched with another controller. When running with the new controller there were no scrap issues and no variation in temperature on the monitoring thermocouples was observed. To verify that the controller was the issue the original controller was again used to run the mold, which in turn resulted in high scrap and excess temperature deviation on the monitoring thermocouples.

Hot runner cables are another area that can create problems. Something as simple as having a wrongly wired cable can lead to problems. The cable must be wired such that each zone is correctly attached to the pin at each end of the cable; in other words pin 1 on plug 1 must be wired to pin 1 on plug 2. Cables can be swapped out to verify that they are not the root cause, and if the cables are swapped and the problem is resolved the original cable should be checked for continuity from one plug to the other.

Hot runner plugs, as shown in Figure 17.2, are assembled with contact pins on the end of all of the wires. These pins can be pushed back in the plug, which will cause issues with contact when the plug is attached to mate. Also take time to verify that the plug actually has pins for each zone of anticipated control. It is possible for a plug to only be wired with fewer zones than its full capability resulting, for example, in a plug that has capacity for eight zones but only has six zones wired.

Figure 17.2 Typical hot runner plug

17.3.1.3 Molding Process: Residence Time

When plastic is allowed to sit for excess time at melt temperature the risk of thermal degradation is elevated. The length of time that the plastic spends in the barrel of the molding machine is referred to as residence time. The material being molded will determine the maximum allowable residence time, and some materials such as polyethylene and polypropylene are much less sensitive to degradation from long residence time especially when compared to something like a PVC or acetal. Figure 17.3 shows purged material that has been degraded because of excess residence time; if this material had been injected into the mold the part would have had brown streaks and splay.

Figure 17.3 Degraded material in purge

The best method for determining the true residence time within the melt delivery system is the use of tracer pellets of another color added at the feed throat. After adding the tracer pellets, continue to mold parts and time how long it takes before the new color shows up in the molded parts.

There are less accurate ways to estimate residence time such as calculating the shot weight versus the barrel shot capacity. When doing this it is important to remember that barrel shot sizes on molding machine are based on polystyrene, which has a density of 1.04 g/cm^3, but when using something other than polystyrene it is important to account for the different density. It is also important to understand that solid density is different from melt density, making the calculation less accurate. Also, the plastic that is retained in the flights of the screw must be accounted for and this is usually done using a multiplier (normally 40%).

17.3.1.4 Molding Process: Back Pressure

If processing with excessive back pressure it is possible to degrade a material, which can in turn lead to brown streaks. Typical back pressures of 1000–2000 psi of plastic pressure are normally used. Evaluation of higher back pressures may be required; sometimes high back pressures are used to improve mixing of color concentrates but this may lead to degradation of the material.

17.3.1.5 Molding Process: Recovery Speed

Running high recovery speeds can generate excess shear, which in turn can overheat the plastic. The ideal recovery speed should be set so that the screw recovers 2–3 seconds prior to mold open. If the screw is recovering very quickly the excess shear and residence time at melt temperature may lead to brown streaks. Always keep in mind that shear is the key driver of the melting of the plastic and excess shear can overheat the plastic.

17.3.2 Brown Streak Troubleshooting Mold Issues

The main mold-related causes of brown streaks are:

- Thermocouple location
- Manifold leaks
- Lubricants
- Bleeding of mold saver
- Hot runner insulator gap
- Sprue bushing seat

17.3.2.1 Mold: Thermocouples

The thermocouples in a hot runner are the key components to maintaining proper temperatures. If the hot runner manifold does not have enough thermocouples it will be impossible to have uniform heat distribution in a hot runner manifold. On long hot drops it is also key to have good thermocouple distribution. If only one thermocouple is covering an excess length of manifold or drop there will likely be areas that will run much hotter than the set point, which can lead to overheating of material and brown streaking. It is also important to ensure that thermocouples are properly seated in the thermocouple well to get an accurate temperature reading.

17.3.2.2 Mold: Manifold Leaks

If a hot runner manifold or drop is leaking material there will be a trap area for material to collect in and degrade. These leaks can allow trapped degraded material to contaminate the melt stream and create brown streaks. Unfortunately, the mold must be pulled to examine the hot runner manifold if there are suspected leaks. One sign of a hot runner leak can be when the process is losing cushion; if the process is losing cushion the plastic is going somewhere, either leaking at the nozzle, leaking past the non-return valve, or leaking in the hot runner manifold. Running a mold with a hot runner leak will eventually lead to failure of heater zones and require a great deal of work to repair.

17.3.2.3 Mold: Lubricant Related Issues

Sometimes brown streaks on molded parts can come from grease contamination on the mold surface. The grease that is used to lubricate the mold components can show up on the surface of the parts and appear as a brown streak. Sometimes when this is the cause you can follow the streak back to a location on the mold that has a greased surface such as an ejector pin, lifter, or a slide. Over-greasing a mold can create a great deal of scrap, so care should always be taken to ensure that components are greased but not over-greased. Be especially careful to never spray the ejector half of the mold with mold cleaner because this will thin the grease and cause it to bleed into the cavity for an extended time period. Even mold savers with high solvent levels may cause greases to degrade and become more prone to bleeding. Always keep a clean mold!

Another factor in the mold that can lead to brown streaks on the molded parts is bleeding caused by mold saver getting trapped in between mold inserts. Certain mold savers can bleed from these insert lines and carry dirt and rust into the mold cavity. If this is the cause of the brown streaking the streaks will likely lead back to the witness lines from where the insert is located. Inspection of the mold may well show dark lines along these insert lines. This is another case where spraying with mold cleaner may cause additional contamination to bleed from the insert.

Case Study: Bleeding at Insert

In this case brown streaks were appearing on the part near an insert seam line. The initial thought on this issue was that there was a water leak, but examination of the mold showed that there was none. During the tooling examination an oily substance was detected on the mold surface. It was determined that the “wet” mold saver used was seeping into the insert seam line and then bleeding into the part. Switching to a dry mold saver solved this issue.

17.3.2.4 Mold: Hot Runner Insulator Gap

Many hot runner tips have an insulator gap that helps to isolate the hot runner heat from the cavity. If these gaps are not sealed (most hot runner manufacturers supply seals to fill these gaps) material will become trapped and can potentially degrade leading to brown streaks. These gaps can also create major issues when conducting color changes.

17.3.2.5 Mold: Sprue Bushing Seat

If the sprue bushing seat on the mold is damaged or does not match the radius on the nozzle tip, leakage may occur, which can lead to brown streaks. Inspecting the sprue bushing seat will show if there is any damage that could cause leaks. If damage is found the seat can easily be remachined to eliminate the problem. In Figure 17.4 an example of a leak between the nozzle tip and the sprue is seen; this type of leak can lead to black specks and brown streak contamination. Please note that cardboard between the nozzle tip and sprue bushing does not seal a leak as much as some processors wish it would.

Figure 17.4 Contamination leakage between nozzle and sprue bushing

17.3.3 Brown Streak Troubleshooting Machine Issues

The main machine-related causes of brown streaks are:

- Heater band
- Thermocouple
- Machine performance
- Hang up area
- Screw design

Machine temperature control is critical to avoiding material degradation. The three main components of temperature control of a machine are the heater bands, the thermocouples, and the controller. Any of these three items can lead to problems with material overheating and degrading.

17.3.3.1 Machine: Heater Bands

The first part of the machine temperature control is the heater bands on the machine. All bands must be working to provide even distribution of the heat to the barrel. If a zone is burned out the resulting lack of temperature control will create a location on the barrel that could result in material degradation. Some machines have heater band burnout detection to automatically detect when a heater has burned out. If you need to verify heater bands on a machine without burnout detection the best method is to use an "Amp Clamp" style meter to verify that the zone has electrical current flowing to it. If the heater is burned out there will be no current present. When replacing heater bands make sure to use the correct width and wattage. Also, heater bands must be mounted tightly to the barrel because if they are loose there will be poor heat transfer.

17.3.3.2 Machine: Thermocouples

Thermocouples are responsible for providing temperature feedback for the process. If the thermocouple is not providing accurate information the temperatures will also not be accurate. The first thing that can go wrong with thermocouples would be using the wrong type, so if the machine is supposed to have a type "J" thermocouple do not replace it with a type "K". Another problem that can occur with thermocouples is when they are not inserted correctly into the thermocouple well on the barrel or nozzle. If the thermocouple is not located to the correct depth it will provide a lower temperature reading than it should and will tell the controller to continue to heat when it should not be heating.

17.3.3.3 Machine: Machine Performance

See Chapter 8 on machine performance.

17.3.3.4 Machine: Hang Up Areas

Brown streaking and other forms of degradation can come from hang up areas within the screw, barrel, end cap, nozzle, or tip. If material is trapped at melt temperature in hang up areas it may degrade and lead to brown streaks. Always make sure that the melt flow channels are as smooth as possible, as any mismatch can result in scrap. Other factors that can cause a hang up at the nozzle tip include damage to the tip or sprue bushing, misalignment of the injection unit, or using the wrong radius tip.

Figure 17.5 shows the seat surface of the nozzle and nozzle tip interface; these should seat out without a gap that will trap material. Also clean this seat surface when replacing a nozzle tip. Another factor to watch for is using too much anti-seize when replacing a nozzle or tip, as excess anti-seize will leach into the melt stream and cause brown streaks.

Figure 17.5 Seat surface between nozzle and nozzle tip: clean when installing

Case Study: Nozzle Tip Hang up

In this case the ABS part was having scrap issues with brown streaks and splay. The location of the defects was consistent and seemed to point to somewhere near the front of the barrel or the hot runner. Examination showed that the nozzle tip was very undersized to the bore opening on the hot runner, which resulted in a ledge that allowed material to catch at the bore opening and degrade. When the nozzle tip was replaced with a proper orifice tip the scrap was eliminated.

17.3.3.5 Machine: Screw Design

Any given screw geometry is not going to be the correct geometry for every material. For materials that are shear sensitive high compression ratios may lead to degraded material from the excess shear. The compression ratio of the screw is the difference between the flight depth in the feed zone of the screw and the flight depth in the metering zone of the screw. Compression ratios for thermoplastic injection molding usually range from 1.5 : 1 to 4.5 : 1 with "GP" screws usually falling between 2.5 : 1 and 3 : 1. The higher the shear rate the more likely shear-sensitive materials will degrade. Many amorphous materials will react better with low compression ratios.

The screw L/D can impact material degradation and lead to brown streaking. L/D refers to the screw flighted length over the diameter of the screw. Larger L/D ratios can provide more gradual application of shear to the plastic. Although a larger L/D (greater than 20 : 1) provides a more gradual introduction of shear it can also lead to longer residence time that could cause brown streaking.

There are a variety of mixing elements available for injection molding screws. Many of these mixing elements help create melt uniformity and improved color mixing. There are mixing elements that will lead to high shear rates and may degrade the material leading to brown streaks.

Case Study: Brown Streaking from Screw Design

In this case study a machine was running a PC/ABS material with a colorant concentrate added at 4%. The job was running well but another screw manufacturer wanted to evaluate their mixing screw in our process. The original screw was replaced with the new screw and immediately scrap from brown streaking started. A variety of process adjustments were made working with the new screw, but nothing seemed to resolve the problem. After running for about 2 weeks with excess scrap the screw was finally replaced with the original screw, which resolved the scrap problem. Examination of the new screw showed that the root of the screw flights seemed to have gotten very hot (steel changed color). It appeared that the mixing zone on this screw was creating way too much shear. The screw designer had some ideas on changing geometry but they ended up missing their opportunity, because no one wanted to continue trying to "fix what wasn't broken", so the original screw stayed in the machine.

Processors should at least understand what type of screw and what the key numbers are for the screw in the machine of choice. Decisions can be made about whether the screw is suitable for the material to be molded. Keep in mind that the term "GP" is a relative term and one "GP" screw is not necessarily the same as another.

When pulling a screw look for hang up areas and locations of carbon buildup on the screw. These will indicate potential areas that will lead to brown streaking (or black specks and splay). See Figure 17.6 for an example of a screw with severe buildup that was leading to molding problems.

Figure 17.6 Screw with buildup of carbon at root of flights

17.3.4 Brown Streak Troubleshooting Material Issues

Some of the factors that can influence brown streaking related to materials include:

- Wrong material
- Contamination
- Color concentrate

17.3.4.1 Material: Wrong Material

If someone accidently loads the wrong material into the hopper, brown streaking may be the result. Because all materials have different melt and degradation temperatures adding the wrong material can result in having excess barrel temperature settings that will lead to degraded material. This could also be dangerous because certain materials will release hazardous materials when they are allowed to degrade. Always ensure that the correct material is being used in the process.

17.3.4.2 Material: Contamination

Plastic materials can be contaminated with a wide variety of foreign materials. These contaminants can include other plastics, dirt, cardboard, food, and a number of other items. These contaminants can all burn at process melt temperatures and

result in brown streaking. Usually when these types of contamination occur they will be short lived because it was most likely an isolated case, for example if someone spilled something in a gaylord of material.

Material can be contaminated during the grinding process. On jobs where regrind is allowed precaution must be taken to ensure that the regrind does not become contaminated with another material. Setting up a robot or sprue picker to automatically drop the runner in the grinder is a good way to help stay consistent when using regrind. Some shops will not allow an operator to grind any parts so that the risk of contamination is minimized.

Residue from a darker color will often cause dark colored streaks. Thorough cleaning and purging during changeover are the key to success.

For more information see Chapter 10 on purging.

17.3.4.3 Material: Color Concentrate

Brown streaking can sometimes link back to the color concentrate being used for the material. If the carrier used for the concentrate cannot withstand the same process temperatures as the base plastic the result could be brown streaking from degradation of the carrier resin. It is also possible that the color concentrate is not mixing well with the material, which may result in streaking that can look like material degradation. Also verify that the color concentrate is being added at the correct let down ratio, because too high of a colorant ratio can result in a variety of appearance issues including streaking.

18 Bubbles

18.1 Description

Bubbles are defects that appear when gas is trapped in the melt and injected into the cavity leaving a cosmetic defect in the part. See Figure 18.1 for an example of bubbles.

Also known as: gas traps, blisters

Mistaken identities: voids, unmelts

Figure 18.1 Bubbles

18.2 Bubbles Troubleshooting Chart

Table 18.1 shows the bubbles troubleshooting chart.

Table 18.1 Bubbles Troubleshooting Chart

Molding Process	Mold	Machine	Material
high melt temperature	venting	machine performance	moisture content
low back pressure	hot runner temperatures	crack in feed throat leaking water	transparent materials
high decompression	cracked water line	screw design	contamination
	Venturi effect		unmelts

18.3 Bubbles Troubleshooting

The first and most important step of troubleshooting this defect is to determine if the issue is a bubble or a void (see Chapter 45). The troubleshooting steps will be different depending on which problem is occurring.

The difference between a bubble and a void is that a bubble is formed by gas in the melt stream whereas a void is a spot where plastic shrink has occurred creating a vacuum in the plastic wall stock. Voids will typically occur in thick sections where the part was not completely packed out. A method to determine if the defect is a bubble or a void is to slowly heat the area with a torch or a heat gun: if the defect is a void the wall stock will collapse and show a sink, but if the defect is a bubble the wall stock will swell as the gas is heated and expands.

Some visual clues to help distinguish a bubble versus a void include the following:

- If the defect is occurring only in thick areas of the part it is most likely a void.
- Bubbles can appear throughout the part and the part may also display signs of splay.
- Voids often times will show signs of sink on the wall.
- If you slowly heat the area a bubble will expand but a void will sink.

When troubleshooting bubbles, it can be very helpful to run a series of short shots. The short shot series will be very useful to establish where specifically the bubble is beginning to appear. If it shows up only in specific locations the starting point of the bubble formation can often point to the root cause.

18.3.1 Bubbles Troubleshooting Molding Process Issues

The main causes of bubble defects in molded parts come from excess gas in the part. This excess gas can come from several root causes including:

- High melt temperature
- Low back pressure
- High decompression

18.3.1.1 Molding Process: High Melt Temperature

When plastic is overheated often a gas will be given off from degradation of the components of the plastic. Much like moisture these gases can form bubbles that can be carried in the melt stream and injected into the cavity of the mold. When troubleshooting an issue such as bubbles it will be beneficial to conduct a melt temperature check. If the melt temperature is above the recommended value for the material being molded it will be time to check the following:

- Barrel temperature set points correct? Verify settings to those recommended for the material.
- Barrel temperatures running to set point? Sometimes a zone will be overheating due to a thermocouple problem such as poor contact between the thermocouple and the steel, a bad thermocouple, a miswired zone (zone 1 thermocouple actually is located on zone 2), or a thermocouple that is located too far away from the heat location.
- Back pressure set or running excessively high? The majority of the heat required to melt the plastic comes from shear through back pressure and recovery rate. Verify that the back pressure is set correctly and giving the correct pressure. Also remember that the intensification ratio will contribute to the actual pressure that the plastic is seeing.
- Screw recovery speed running excessively high? Ensure that the screw RPMs are both set correctly and not running excessively fast. Under normal circumstances the screw should recover 2–3 seconds prior to mold opening.

Often times when overheating occurs it will be caused by either an incorrect setting on the machine or something running out of control. Overheating can lead to many other issues including splay, brown streaks, or black specks. You may see a combination of these defects if overheating is the root cause.

18.3.1.2 Molding Process: Low Back Pressure

If the back pressure is set too low the shot will not be adequately compressed. When the shot is not compressed adequately air voids will be present in the melt stream which will be injected into the cavity potentially resulting in bubbles. Typical back pressure settings for most materials will be between 1000 and 2500 psi

plastic pressure. If running with back pressure lower than 1000 psi plastic pressure the process should be evaluated to determine if the setting can be changed.

Low back pressure will lead to a poor uniformity of the melt. Normally, low back pressures are used for shear-sensitive or glass-filled materials. Verify that the back pressure is set to the documented process.

One commonly occurring mistake is when people are purging a machine they will drop the back pressure to help the screw recover, but forget to reset the back pressure to the set point. With these low back pressures, the air that is located between the pellets in the barrel will not be forced out, resulting in a melt that is full of trapped gas. If lowering back pressure to help with screw recovery make sure to reset the value to the required set point.

18.3.1.3 Molding Process: High Decompression

Decompression is an important setting on a molding machine to help with controlling drool and for helping a non-return valve seat. However, remember that excessive decompression may pull air into the plastic melt, which in turn can lead to bubbles in the molded part.

Verify that the decompression setting is correct. Evaluate if the decompression can be lowered without risking other problems. Avoid the habit of using more decompression than is required for producing a quality part.

If excessive decompression is being used to control drool or stringing problems review the nozzle heater actual values to ensure that temperatures are not the true cause of the problem. There will be cases where dropping the nozzle temperatures will allow the decompression to be reduced, which in turn leads to an elimination of bubbles and drooling problems.

18.3.2 Bubbles Troubleshooting Mold Issues

Potential reasons for bubble defects relating to mold issues include:

- Venting
- Hot runner temperatures
- Cracked water line
- Venturi effect

18.3.2.1 Mold: Venting

The most common mold-related problem that can lead to bubble problems is a lack of venting. If gas is trapped in the melt stream and not allowed to vent properly the gas can very easily lead to a bubble.

See Chapter 7 on venting for more details.

18.3.2.2 Mold: Hot Runner Temperatures

If hot runners are set or running with an excessively high temperature the material may degrade in the hot runner manifold leading to generation of gas. This gas will be entrapped in the plastic melt stream and will be carried into the mold during injection.

Verify that the hot runner temperatures are set correctly as per the setup sheet. It is also critical to determine if the hot runner system is running at the set point. Often times a hot runner zone will overheat due to poor thermal control from causes like a thermocouple being installed incorrectly. If a hot runner system has two thermocouples swapped the hot runner controller will keep increasing the current flow to try to get the zone to achieve the desired set point, but the thermocouple will be reading another location. More sophisticated hot runner controllers have many built-in features such as broken thermocouple detection, swapped thermocouple detection, and heater burnout detection.

18.3.2.3 Mold: Cracked Water Line

Another tooling issue that can cause bubbles is a crack in the mold. If a mold cracks to a water line the water can wick to the surface of the mold resulting in bubbles. Cracks in the mold can be obvious with water drips showing on the mold surface. However, there will be cases where the water only shows up after the mold has been closed and clamped up, so when checking for suspected cracks always close the mold to see if the clamp pressure opens up the crack. A short-term fix for a cracked mold is to use a reverse-flow thermolator, as this can affect the cooling capability of the mold. The ultimate fix will be to fix the crack. Be aware during tool design that sharp radii will be more prone to crack.

Cracked water lines often are a result of a mold design mistake. Sharp corners in steel will be stress risers that can lead to cracking. Also, if a water line is placed too close to the surface of the mold the resulting thin steel condition will often lead to cracking. Examination of mold designs is a critical step to providing a mold that will meet production requirements.

18.3.2.4 Mold: Venturi Effect

The Venturi effect is named after Giovanni Battista Venturi and describes a pressure differential effect that occurs during flow. In injection molding we sometimes see an effect like a Venturi when air is pulled into the melt as the mold is filled.

This effect can come from a variety of areas in the mold including mismatch or poor seating between the nozzle tip and sprue bushing, mismatch or seating problems in the hot runner manifold, and areas in the mold with gaps such as ejector pins and inserts. If the bubble seems to downstream to specific areas of the mold it may indicate a problem with a mold component. This is a problem that will require

more in-depth evaluation to resolve as a root cause. To determine mismatch areas the mold may need to be disassembled and blued up to determine potential areas of concern.

18.3.3 Bubbles Troubleshooting Machine Issues

The main issues that can create bubbles that are related to the machine are as follows:

- Machine performance
- Crack in feed throat leaking water
- Screw design

18.3.3.1 Machine: Machine Performance

See Chapter 8 on machine performance.

18.3.3.2 Machine: Crack in Feed Throat

If there is a crack in the feed throat of the machine it is possible for the feed throat cooling water to enter the barrel along with the plastic pellets. As this water is introduced to the plastic it will turn to steam as it is heated, and the steam gas will then be trapped in the melted plastic which can cause gas bubbles in the part.

To check for feed throat cracks the barrel must be emptied and the hopper removed from the feed throat. Use an inspection mirror to look in the feed throat for any water drips. If a crack is detected it will need to be fixed to resolve the scrap issues.

Extreme caution should always be used when examining a feed throat because pressure in the barrel can blow back through the feed throat. Figure 18.2 shows an inspection mirror for examining a feed throat. A face shield is another safety precaution to be used when dealing with feed throats. Never look directly in or stand directly above the feed throat.

Figure 18.2 Inspection mirror for safely examining a feed throat

18.3.3.3 Machine: Screw Design

If the screw in the machine is of wrong design for the material being molded, it is possible to have gas entrapped in the melt stream. A key to verify the screw design is the compression ratio which is the ratio between the feed flight depth versus the metering flight depth. If the machine has a low compression ratio screw (under 2.5) there may be issues trying to run materials such as PMMA, acetal, nylon, etc.

Note that most machines will have a medium compression ratio screw of between 2.5–3, which will be adequate in most cases. If the machine has a specific low compression screw it could be an issue. Low compression ratio screws are commonly used for molding materials like PVC.

18.3.4 Bubbles Troubleshooting Material Issues

Material-related issues with bubbles may come from the following:

- Moisture content of material
- Transparent materials
- Contamination
- Unmelts

18.3.4.1 Material: Moisture Content

If a plastic material is molded wet the moisture will turn into gas as the material is heated in the barrel. As this gas is formed in the melted plastic it will tend to form bubbles. These bubbles can be trapped within the molded part resulting in defects. All hygroscopic materials must be adequately dried prior to processing.

Moisture content being too high is probably the leading overall cause of bubbles in molded parts. When troubleshooting bubble problems one of the first items to check is the moisture content of the material (see Chapter 9 on drying).

18.3.4.2 Material: Transparent Materials

Transparent materials are not necessarily more prone to experiencing bubbles but rather it is much easier to see bubbles in a transparent part. Clear, hygroscopic materials like polycarbonate, clear polyesters, and PMMA will display bubble defects if molded wet.

When molded, opaque materials can have bubbles that will not be noticed. Due to this fact, parts molded with opaque materials will tend to have less scrap for bubble defects. There may very well be bubbles present that are not detectable. Generally speaking, because bubbles will not be visible in an opaque material, parts will not be scrapped due to bubble problems; however, they may be suffering the same root cause and end up being scrapped for splay.

18.3.4.3 Material: Contamination

If the molding material is contaminated with a foreign material, adverse reactions can occur. In many cases, if a lower melting temperature material is contaminating a higher melting temperature material the lower melting temperature material will likely degrade during the melting process and in turn create volatiles that outgas. These outgassed volatiles will be carried along in the melt stream and in turn may lead to a bubble.

Keys to avoid contamination of material include the following:

- Ensure that containers are covered to reduce opportunities for contamination.
- Clearly identify all materials, additives, and regrind to avoid mixing materials.
- Establish clear guidelines for introducing a new bag, box, or silo feed of material.

Contaminated material may lead to a variety of visual defects, but will also compromise the physical properties of the material potentially leading to parts that will not perform in end-use application.

18.3.4.4 Material: Unmelts

Unmelts are unmelted pellets that make their way through the barrel without melting. They can sometimes be misidentified as bubbles because they often leave a pronounced mound on the part. If unmelts are suspected, cut the part in the area of the bubble to see if there is a pellet under the mound.

19 Buildup

19.1 Description

Buildup is an issue that can occur from outgassing of volatiles from the plastic material being molded. Buildup will often lead to other issues such as splay because buildup in the vents reduces the ability of the mold to vent gas out of the cavity. Sometimes buildup will occur on the cavity of the mold leading to cosmetic defects that appear rough. Figure 19.1 shows a mold with severe buildup at the vents: notice the staining and erosion of the cavity steel.

Also known as: plate out, gas buildup

Mistaken identities: none

Figure 19.1 Severe buildup causing steel erosion

19.2 Buildup Troubleshooting Chart

Table 19.1 shows the buildup troubleshooting chart.

Table 19.1 Buildup Troubleshooting Chart

Molding Process	Mold	Machine	Material
high melt temperature	vents	machine performance	regrind
excess shear			moisture content
			material type
			additives

19.3 Buildup Troubleshooting

The root cause of most buildup issues is volatiles leaving the plastic that in turn collect on the mold surface. To resolve a buildup problem, it is important to address two main questions:

1. What is causing the excess volatiles?
2. Can the volatiles be better vented?

19.3.1 Buildup Troubleshooting Molding Process Issues

There are several process-related factors that can lead to excess volatile byproducts from the molding material. These factors include:

- High melt temperature
- Excess shear

19.3.1.1 Molding Process: High Melt Temperature

Frequently the root cause of buildup issues stems from the material releasing excess gas or volatiles from being degraded. If the material is being processed at a too high temperature it is more likely that excess gas will be generated, which will tend to foul the vents and mold surface quicker. When troubleshooting a buildup problem verify that the material is not being overheated during processing.

The melt temperature should be verified to ensure that the material is not being exposed to higher then recommended temperatures. It is important to measure the actual melt temperature rather than relying on barrel temperature set points.

19.3.1.2 Molding Process: Excess Shear

Shear contributes a great deal of the energy input for melting the plastic. It is important to understand that high levels of shear can be generated during processing, which in turn can lead to excessively high melt temperatures, which can lead to material degradation. Key items to check for high shear include:

- Back pressure
- Screw recovery speed
- Injection fill velocity

To verify the impact of these factors, check the melt temperature of the material. Often when shear levels are high during plastication it will be possible to detect overriding barrel zones. A good check is to examine the barrel temperatures to determine if they are running to the set point; overriding zones may indicate that there is excessive shear as the screw recovers.

When filling a mold with high velocity the gases generated during processing will have less chance to escape through the vents of the mold, which results in faster fill velocities requiring more venting. If fill time seems to contribute to the buildup issues, first try to add additional venting to the mold rather than immediately slow down the fill velocity. Slower fill velocities result in longer cycle times and make the part more expensive to produce. Way too often there are cases where someone slows down fill velocities to eliminate a defect caused by lack of venting (see Section 19.3.2 for more on venting).

One key in processing is to avoid conditions that will damage the material. Shear rates, temperatures, and residence time are the typical process conditions that are most likely to damage a material.

19.3.2 Buildup Troubleshooting Mold Issues

The main issue that contributes to buildup relates to venting. If the mold does not have adequate venting the volatiles from the plastic will be trapped in the mold, which will lead to potential problems with buildup on the mold surfaces. See Figure 19.2 for an example of buildup on the mold surface; notice how the black buildup has been scraped off the mold surface.

Figure 19.2 Severe vent buildup: notice black buildup that has been scraped off the mold

When buildup is creating negative effects on the parts or process it is important to verify that the mold has adequate venting and that all of the vents are open. Often times the buildup will try to escape through the vent and create a large amount of residue in the vent land, which will further restrict the mold's ability to vent. If experiencing venting-related buildup, first verify that the vents are open. To verify that vents are open the mold surface can have tooling blue applied to the side opposite where the vents have been cut. If the vents are open at all the blue will not transfer in the relieved areas of the vent. It can also be critical to know that the vent is deep enough; the depth can be measured with a depth micrometer or by closing up on a putty and measuring the putty thickness across the vent location. If the vents are found to be too shallow they should be opened up by an experienced mold technician.

If the vents are determined to be open the mold should be reviewed to determine if additional venting can be added. Frequently the old standby of placing a ½″-wide vent every "X" inches will not be adequate. Rarely will too much venting create an issue (unless the vents are too deep and flash). Often times a full perimeter vent will yield excellent results and allow the gas to easily escape the mold. If full perimeter venting cannot be used an approach of using wider vents will often work well. A situation that is common is trying to hit an end of fill location with a ½″-wide vent, so the odds of getting everything perfect are reduced, and if a 2″-wide vent was used the window to hit the end of fill location is much wider.

Another factor that will have a large effect on buildup is the land length of the vent. If the land length is too long the vents will not stay as clean and will plug with buildup much quicker. Land length on vents should be kept below 0.080 in. Often

times just reducing an excessive land length will improve a buildup issue. Also be aware that if the vent relief channel has not been squared to the mold edge you will only have the desired land length at the center of the radius from the cutter.

Applying diamond chrome with PC materials can help with buildup. Other mold coatings may help to minimize buildup and improve mold cleaning.

In some cases with buildup, adding vacuum to the vents can improve the overall ability of the mold to vent. There are ways to do this with seals at the parting line but in some cases just adding a vacuum line to the vent relief can provide good results. Connect the vacuum line to a vacuum generator and use a mold closed switch to actuate the vacuum to draw the volatiles out of the vents.

19.3.3 Buildup Troubleshooting Machine Issues

The main machine-related cause of buildup is machine performance.

See Chapter 8 on machine performance for details.

19.3.4 Buildup Troubleshooting Material Issues

Possible material-related problems include:

- Regrind
- Moisture content
- Material type
- Additives

19.3.4.1 Material: Regrind

Use of regrind can contribute to buildup, especially in a case where the regrind has been degraded or is molded wet. Anything that causes an increase in the amount of gas generated during material plastication can lead to problems with buildup.

Note that there are also cases where using regrind may help reduce a buildup problem because some of the components leading to buildup have been burned out with the extra heat history. Things like low molecular weight resins, release agents, residual monomers, or other additives may actually be reduced after the first pass through the molding machine. Regrind can have a significant impact on the process and a good way to determine the impact is to mold 100% virgin material and evaluate if the problem changes.

Regrind can be a source of contamination. If a part or runner of another material is dropped into a grinder the result will be contaminated regrind. Contaminated regrind could degrade due to differing melt temperatures, which then can lead to buildup. See Figure 19.3 for an example of contaminated regrind. If contamination is suspected take some material and spread it out across a table sorting through the pellets looking for contamination.

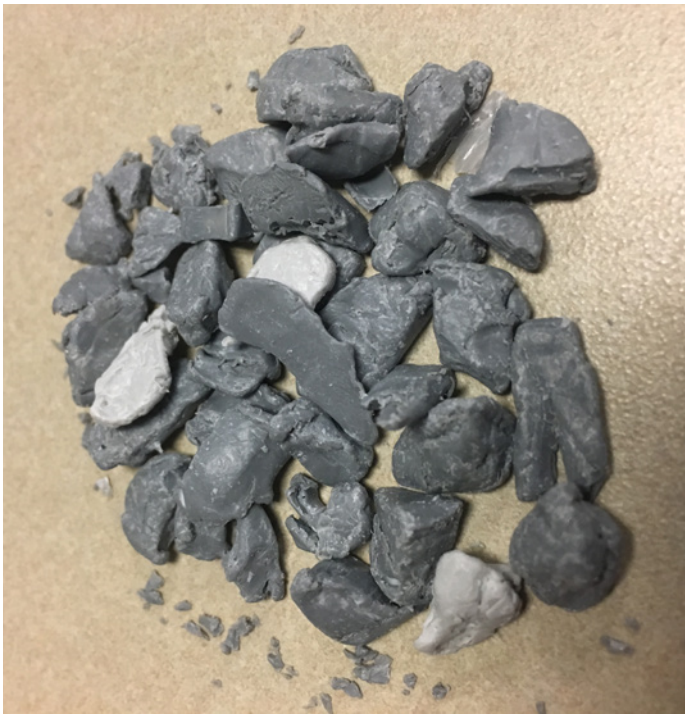


Figure 19.3 Contaminated regrind: notice light gray material

19.3.4.2 Material: Moisture Content

When plastic is processed without being adequately dried the moisture will volatilize in the barrel of the molding machine producing gas. This extra gas generated during the melting process will need to be vented from the mold while the mold is being filled. The venting on the mold may not be adequate to handle the extra gas that is being injected during fill. As this gas tries to escape the cavity via the vents buildup may be left behind in the vents. Hence, ensure that the material has been adequately dried prior to processing. For more details on drying of materials see Chapter 9.

19.3.4.3 Material: Type

There are some materials, especially wide-spec materials, that will tend to have more issues with buildup. In some cases, these materials may have components and monomers that are not fully reacted which end up being volatilized during the melting process. The volatiles will then end up trying to escape the mold through the vents, where these residual byproducts will then leave buildup.

An example would be caprolactam in nylon. Sometimes this unreacted caprolactam will result in a white buildup on the mold and/or part surface. This buildup in vents will lead to burning of the part, and the white buildup on the part will be a visual defect. Due to a wide range of moisture content in nylon and its low viscosity it can be challenging to provide adequate venting to allow all volatiles to escape while the mold is filling.

Polycarbonate can be another material that tends to form buildup while running. Polycarbonate is more forgiving than many materials for vent depth so it is often easier to get adequate venting with polycarbonate. Evaluate the mold per the recommended venting guidelines.

Many ABS and PPO materials will also have volatiles that will result in a liquid that builds up on parting lines.

Adequate venting is critical to successful molding with all materials. If the material being molded seems to be causing more issues than normal make sure to provide optimized venting.

19.3.4.4 Material: Additives

Plastics are produced with a wide range of additives that can migrate out of the material during processing. Additives that may contribute to buildup on the mold include:

- Flame retardants
- Colorants
- Lubricants
- Release agents
- UV stabilizers

There will be cases where the additive package used in a material will lead to buildup problems. Unfortunately, these additives may be necessary to provide the plastic with specific required properties. If a particular material is causing an ongoing issue verify that all processing recommendations are being followed and verify that venting is maximized (see Chapter 7 on venting).

When using any additives as a masterbatch it is important to verify that the required let down ratio is being used. If an excess amount of an additive is being

used in a material there is a good chance that byproducts from the additive will plate out during the molding process.

There will be buildup cases where the material supplier should be consulted. The material supplier will sometimes ask for a sterile swab of the buildup material. The swab will then be sent to the lab for analytical studies to determine what the chemical composition of the buildup is.

Case Study: Buildup

A two-cavity mold was running a super-tough nylon and experiencing severe buildup problems. The mold had to be stopped and cleaned every 2 hours to avoid complete buildup of the vents which in turn led to burning of the part. This problem was relieved by adding additional parting line venting; this allowed the mold cleaning interval to go to once a shift (instead of four times per shift). The nature of this material required additional venting surface area to improve the buildup of byproducts in the vents. As the material was a nylon the vent depth had to be kept shallow to avoid flashing, so extra vent width was added to allow better venting.

20 Burns

20.1 Description

When gases are trapped during the filling of a mold, the high pressures may cause ignition to occur causing the plastic to burn. This defect will often show up as black smutty deposits on the part and the mold surface. See Figure 20.1 for an example of a burn defect.

Also known as: dieseling, gas traps

Mistaken identities: color swirl, brown streaking

Figure 20.1 Burn

20.2 Burns Troubleshooting Chart

Table 20.1 shows the burns troubleshooting chart.

Table 20.1 Burns Troubleshooting Chart

Molding Process	Mold	Machine	Material
too fast fill velocity	venting	buildup on screw, end cap, etc.	moisture content
high melt temperature	grease, oil, or other contamination	screw design	type
high back pressure		barrel heater control problems	additives
decompression		damage to screw, barrel, tip, etc.	

20.3 Burns Troubleshooting

Burns are areas of the molded part that have a brown or black discoloration that is often sooty (like charcoal). The two main questions when troubleshooting burns are:

1. Where is the gas coming from? Keep in mind that the cavity is full of gas (air) at the start of fill.
2. Why is the gas not escaping the mold?

Burn troubleshooting often comes down to venting-related issues; do not process around tooling issues!

20.3.1 Burns Troubleshooting Molding Process Issues

As mentioned above, the root cause of burns is frequently a lack of venting. Processing around a burning issue should be the last choice; instead address tooling issues before adjusting the process!

Process settings that can either create excess gas byproducts or cause the gas to be trapped include:

- Fill velocity
- Melt temperature

- Back pressure
- Decompression

20.3.1.1 Molding Process: Fill Velocity

With fast fill speeds it is more likely that the gas will be trapped rather than escaping through the vents. Often times slowing fill velocity will improve a burning issue; however, this is a compromise on cycle time. Typically, if slowing the fill velocity improves the burn defect improving the venting on the mold will also fix the problem (fix the root cause). If burns are a problem venting should always be reviewed prior to modifying the molding process.

Slowing the fill speed will not only increase the cycle time but may also impact the viscosity of the material as it is injected into the mold. This viscosity shift may in turn lead to additional part defects such as uneven gloss, sink, short shots, etc. Slowing the fill speed on an existing process to compensate for another root cause is not acceptable, so determine the root cause and address the issue.

Do not process around a venting issue! See Chapter 7 for more information on venting.

Case Study: Fill Velocity

This was a case with a burn appearing at an end of fill knit line. The burn had the classic appearance of charred material and was leaving a black deposit on the surface of the mold in this area. Slowing the velocity eliminated the burn by giving the gas time to escape the cavity. A mold maker was brought over and he opened the vent to a deeper depth, which allowed the fill velocity to be set at the original value. By opening the vents it was possible to achieve a faster fill time, which saved over 2 seconds of cycle time.

20.3.1.2 Molding Process: Melt Temperature

When processing at high melt temperatures it is possible to degrade the material and create gas byproducts that will lead to additional gas that may become trapped in the cavity. Verification of actual melt temperatures should be carried out if the above steps show no impact on results. Always be aware of the recommended melt temperature for the material being molded. All process personnel on the shop floor should have access to the recommended processing conditions for the materials being used.

With melt temperature concerns it is important to verify the factors that influence the actual melt temperature, which are barrel temperatures, back pressure, and screw rotational speed. Do not assume that the barrel temperature settings are the actual melt temperature.

If barrel temperatures are set too high it is easily possible to overheat a material and potentially degrade components of the material. Verify that the barrel temperatures are set correctly and are running to the set point. Look for cases where a heater zone may be calling for power 100% of the time as this may indicate that either the heater band is not adequately heating or the thermocouple is not reading correctly. Keep in mind that heater bands must be mounted tight to the barrel and thermocouples must be located at the correct depth in the thermocouple hole.

Because shear provides much of the energy required to melt the plastic it is important to check back pressure and screw rotational speed when verifying high melt temperatures. Just like with barrel temperatures it is possible that the back pressure or the screw speed was set incorrectly, so always verify to the specified value on the set-up sheet. The general rule of thumb for recovery speed is that the screw should reach its shot size 2–3 seconds prior to mold opening. Back pressure will normally be run in the 1000–3000 psi range depending on material and colorant package.

20.3.1.3 Molding Process: Decompression

Excessive decompression can lead to air being pulled into the melt at the nozzle tip. When this air is injected into the mold with the plastic it must be vented. The extra gas that the decompression pulled into the shot can be enough to overwhelm the venting and result in a gas trap and burn.

Verify that excessive decompression is not being used. If a large decompression is used to limit nozzle drool or stringing investigate the nozzle temperature. Excess nozzle temperatures can lead to drool and require higher than normal decompression settings. Try lowering the decompression setting to determine impact on the process; remember to keep enough decompression to allow the non-return valve to function correctly.

20.3.2 Burns Troubleshooting Mold Issues

When dealing with burning problems the mold is very often the culprit. The following mold-related concerns can contribute to burns:

- Venting
- Grease, oil, or other contaminants

20.3.2.1 Mold: Venting

The number one root cause of burning is lack of venting in the mold! When burning is a problem always start with a review of the venting on the mold. Improved venting will eliminate most burning issues (see Chapter 7 on venting). A classic example of burning from lack of venting at a knit line is shown in Figure 20.2.

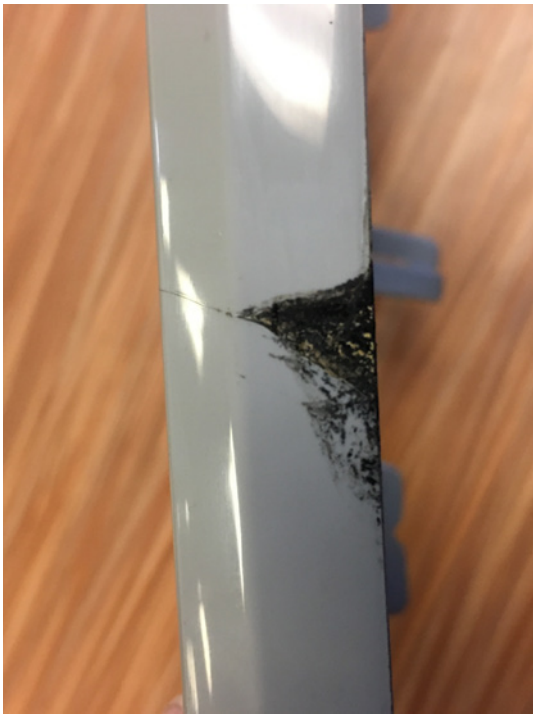


Figure 20.2 Classic example of burning from lack of venting at knit line

When a mold is dirty venting will be compromised. One of the first things to do if a burning problem appears after a mold has been running good parts is to clean the mold. If cleaning the vents of the mold (make sure to check for venting on slides and lifters) solves the problem then the problem is definitely a venting-related issue. To completely eliminate this problem, it may be necessary to add additional venting or plan for more frequent mold cleaning during production runs. If the mold is requiring frequent cleaning of the parting line and vents it is a good indication that the mold needs additional venting.

If a mold is prone to burning there is a high risk that the steel in the burn area will start to erode over time. This erosion of steel can lead to many issues including sticking parts, poor appearance, and parting line flash. It is better to address the burning problem than to repair the mold for erosion-related issues.

20.3.2.2 Mold: Contaminants

If a mold has surface contaminants such as grease from moving components or oil from cylinders it is possible that this contaminant may trap in locations of the mold including vents. If this contamination is an ongoing issue determine what the cause of the contamination is and have it fixed.

When a mold has a hydraulic leak, it is usually possible to detect oil dripping somewhere on the mold surface or from the cylinder itself. When this oil is carried into the melt stream the contamination can lead to plugged vents. If a cylinder is leaking oil it may need to be rebuilt or replaced. There can also be cases where a hydraulic connection is located on the top of the mold, and when the fitting is unplugged there can be drips of oil that are left behind and will often drip down into the mold cavity. This is a good reason to keep hydraulic connections off the top of the mold whenever possible. Also make sure that standard operating procedure for technicians and mold setters is to wipe off any leaked oil.

Excessive grease on moving components can end up in the mold cavity and result in problems including plugged vents. Frequently when a mold comes back from service it will be "over-greased" as mold shops will have a tendency to go with the approach of if a little grease is good a lot is better. Work with your tool shops to establish guidelines for the appropriate amount of grease to be used on a mold. Also, anyone in your shop who applies grease to a mold should be familiar with proper application of grease on mold components.

20.3.3 Burns Troubleshooting Machine Issues

Verify machine conditions to ensure that the machine is not running out of control on temperature, back pressure, or recovery speed. Any overheating of the material may lead to issues with excess gas that can in turn lead to burning.

The main machine-related causes are:

- Buildup on screw, end cap, check valve
- Wrong screw design for material
- Overriding barrel heaters
- Damage to screw, barrel, check valve

20.3.3.1 Machine: Buildup

Buildup is a condition on the machine where material is collecting and degrading over time. As this material degrades it can release additional outgassing and by-products that can be difficult to vent in the mold. If these additional gases are not vented from the mold they could become a gas trap that in turn leads to a burn. Buildup can often become an issue when a machine is allowed to sit idle with material in the barrel, or when transitioning between materials.

Frequently when buildup is a problem there will be other symptoms including black specks, splay, and brown streaking that appear on the molded parts. Thorough purging at material changes or after extended downtime will help minimize this issue.

Sometimes buildup issues will come from a location of mismatch between the nozzle and end cap or the nozzle and tip. It is good practice to have a free flow path as the material moves through the front-end components of the machine. Any mismatch creates a place where material can be trapped and degrade over time because it will have a long residence time.

Proper shutdown procedures are critical to reduce the chance for buildup issues. When a machine is to be down for any amount of time it should be purged to eliminate the chances of degrading the material in the barrel. With some materials this becomes more critical because the more temperature sensitive a material is the faster it will degrade leading to potential buildup problems.

20.3.3.2 Machine: Wrong Screw Design

Shear imparted on the pellets provides much of the energy to melt the pellets as they travel the length of the screw. If a screw is designed without the correct length/diameter (L/D) ratio or the wrong compression ratio it is possible to generate excess shear, which in turn can lead to degradation of the material, which will create excess gas byproducts that must be vented from the mold. A typical general-purpose screw is shown in Figure 20.3.

Figure 20.3 Typical general-purpose screw

Keep in mind that if the mold has been running well in this machine and the screw has not been changed the cause is probably not related to the screw. There can be cases where degraded material builds up on the screw and leads to conditions where flight depths of the screw are different due to the buildup on the root of the screw. Also, in extreme cases a screw can have so much buildup of carbonized, degraded material that the screw looks like it has a barrier flight from the buildup. If this is causing problems it is critical to establish proper shutdown and purging procedures.

20.3.3.3 Machine: Overriding Barrel Heaters

As with excessive shear an area of overheating on the barrel can lead to problems with degrading of material. If there is a zone on the barrel, end cap, or nozzle that is running above the set points it could lead to a spot that is overheating and degrading the plastic.

Examination of the barrel temperature readings will indicate if there are zones that are overheating. Often the zone may actually be overriding from the shear heating during screw recovery. If the zone is running over the set point but not actually calling for heat one of two possibilities exist:

1. The shear created from the rotating screw is generating heat and higher temperature than the set point and the barrel heaters are no longer being used to control the barrel temperature at all. If this is the case look at reducing the screw rotation speed to allow the screw to recover 2–3 seconds prior to mold open, or examine the back pressure being applied to the screw.
2. There is a remote possibility that the thermocouples have been installed in the incorrect locations. This is more likely a potential cause if heaters or thermocouples had been recently replaced on the machine. If this is the case you will see a zone that is running hot without calling for heat and another zone will be running cold while calling for heat continuously. This is a good indicator that the thermocouples have been cross wired.

   IR photos can be used to discover hot spots in the nozzle as shown in Figure 20.4.

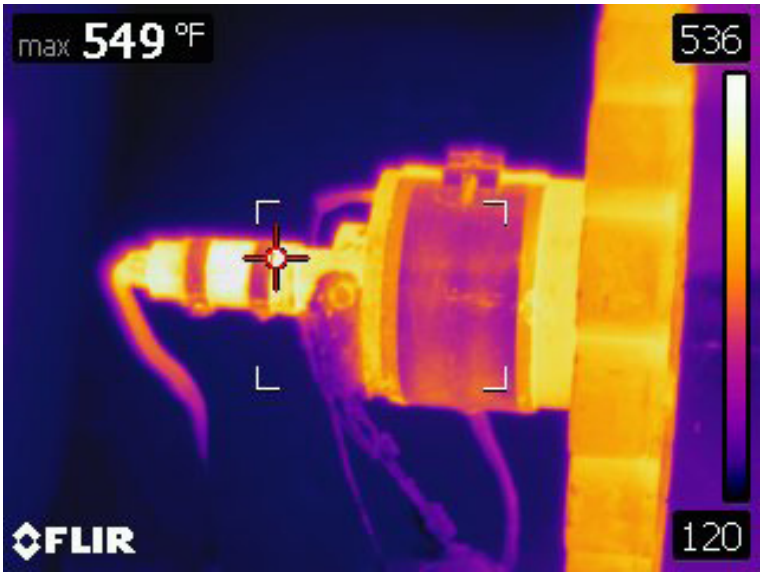


Figure 20.4 IR photo of nozzle looking for hot spots

20.3.3.4 Machine: Damaged Components

Damaged components in the screw, barrel, check valve, end cap, nozzle, or tip can lead to areas of high shear and or buildup locations. The damaged components can

create additional outgassing from the plastic that will in turn require additional venting to be released from the mold. Sometimes, rather than damage, the issue may be metal that is trapped somewhere along the feed system.

Unfortunately, to check for trapped metal or damage, the end cap must be pulled from the machine and each of the components inspected. This is a very time-consuming endeavor and should be undertaken only when confident that there is trapped metal or damage in the components of the injection unit.

20.3.4 Burns Troubleshooting Material Issues

Factors based on material issues that can contribute to burning include:

- Moisture content
- Type
- Additives

20.3.4.1 Material: Moisture Content

Burns will normally be a result of volatiles in the melt stream not escaping while the mold is filled. When hygroscopic materials are molded with excess moisture content the water will vaporize during plastication and put gas into the melt stream. This gas in the melt stream will normally appear as a splay related defect, but can lead to burning as it tries to travel through the vents in the mold. See Chapter 9 on drying for more information on moisture.

20.3.4.2 Material: Type

As with many defects, some materials will be more likely to burn than others. Very temperature-sensitive materials such as PVC or acetal will degrade much faster and can actually reach a point where the material in the barrel is already in a burnt state. When processing temperature-sensitive materials it is critical to be very aware of the residence time in the barrel and ensure that all of the temperature zones on the barrel and hot runner are accurately controlled with no areas for material to hang up.

As always with any potential material issue make sure that the process is running to the required conditions. Also verify that the correct material is being used and is not contaminated.

Another factor to consider is when conducting a changeover, the barrel temperatures must be allowed to cool sufficiently when switching to a lower melt temperature material. If switching to a temperature-sensitive material like a PVC or acetal make sure that the barrel temperatures have dropped to the required temperatures for the material.

Case Study: PVC Burning

In this particular part an alternate material PVC was being sampled. When the new material was sampled in the mold, there were several spots of charred material showing on the surface. Numerous processing experiments were conducted to minimize the burning of the material. After reviewing the situation with the material supplier, it was determined that the formulation was off and the material was not thermally stable enough to mold. Numerous hours of molding trials could have been saved if the formulation was better understood originally.

20.3.4.3 Material: Additives

Color concentrates can impact the likelihood of burning as well as the odds of detecting the burns. Some concentrates will degrade faster and produce more of an outgassing effect. Also, burns on lighter colors will be much more easily detected, even for small burns. Because dark colors hide burns better it is critical when reviewing a part to examine any deep ribs for possible burning. There are cases where the burning on the tip of a rib is not even noticed on the part until someone rubs some of the black soot off onto their hands; closer inspection in turn shows the burn location.

Other additive components in a material such as plasticizers or release agents can lead to excess outgassing or buildup in the vents. As mentioned in Section 20.3.2.1, venting is critical to avoid burns during processing.

It is important if color concentrates or other additives are added to the base material prior to processing (either batch blended or at the throat) to ensure that the correct letdown ratio is being used. If an excess amount of an additive is mixed in, the likelihood of creating problems in the process will increase significantly. Also verify that the additive or color concentrate is intended for use in the material being molded.

21 Cloudiness

21.1 Description

Cloudiness is a visual defect that occurs on parts molded with transparent materials. The clarity of the molded part will be reduced and there may even be a white tinge or streakiness to the part (see Figure 21.1). In some cases, streaks of white will appear in the clear material.

Also known as: milkiness, contamination, white streaks

Mistaken identities: color swirls

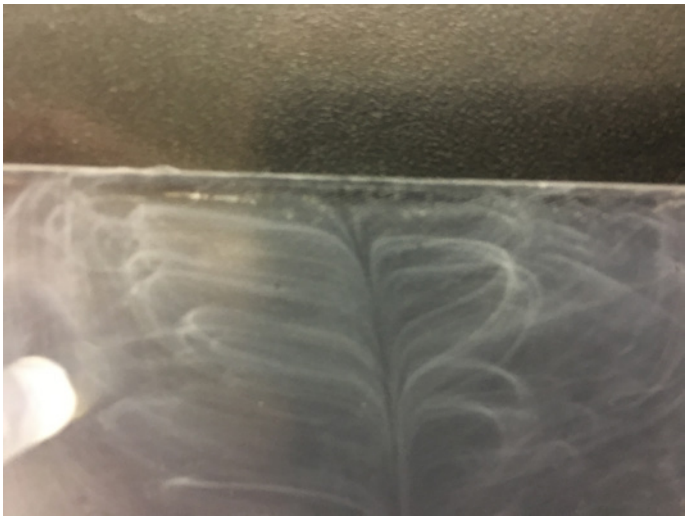


Figure 21.1 Cloudiness

21.2 Cloudiness Troubleshooting Chart

Table 21.1 shows the cloudiness troubleshooting chart.

Table 21.1 Cloudiness Troubleshooting Chart

Molding Process	Mold	Machine	Material
low melt temperature	water leak	hang up areas	contamination
improper material change	surface of mold		semicrystalline nucleation
low mold temperature	hang up areas		moisture content

21.3 Cloudiness Troubleshooting

Cloudiness is most typically an issue with the material, specifically with contamination of the material.

21.3.1 Cloudiness Troubleshooting Molding Process Issues

The key items to verify cloudiness in the process are:

- Melt temperature
- Improper material change
- Mold temperature

21.3.1.1 Molding Process: Melt Temperature

Verify that the melt temperature of the material is appropriate according to the material supplier's recommendations. An actual melt temperature should be measured either with a melt probe or with an IR gun. Actual melt temperature is impacted by a variety of factors including barrel temperatures, back pressure, screw rotation speed, residence time, and screw geometry.

If the melt temperature is running low it can lead to a cloudy appearance. Try raising the melt temperature to see if the cloudy appearance improves.

Sometimes low melt or mold temperatures can lead to a jetting defect that actually produces a cloudy appearance, as shown in Figure 21.2. In this example the jetting is obvious; however, there are times when the jetting effect is the root cause but it is not apparent that jetting is occurring until short shots are run.

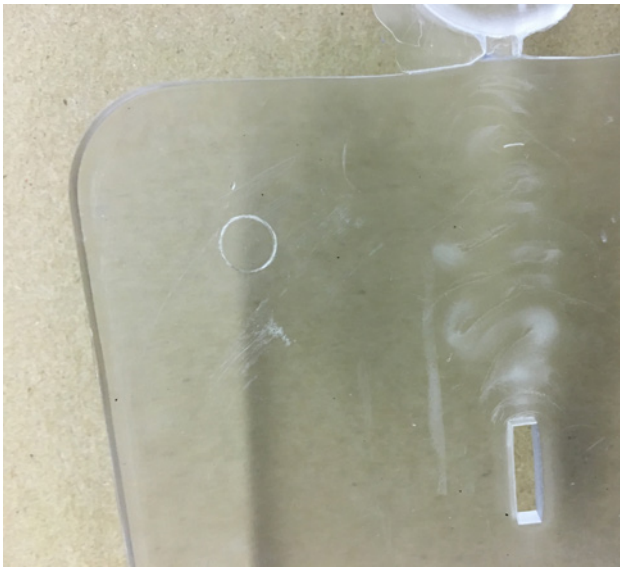


Figure 21.2 Jetting in a clear polycarbonate plaque resulting in a cloudy appearance

21.3.1.2 Molding Process: Improper Material Change

An often-seen cause of cloudiness is a poor changeover. When conducting a material change any residue from the previous material can lead to a cloudy appearance on a transparent material.

When changing to a clear material the material changeover procedure is very important. Any leftover material anywhere in either the feed stream or the melt delivery stream can lead to cloudy parts. Always ensure adequate amounts of the clear material have been purged to ensure an uncontaminated melt stream. Always take the time to conduct a thorough material change.

See Chapter 10 for more information on changeover recommendations.

21.3.1.3 Molding Process: Mold Temperature

A change in mold temperature may impact the cloudiness of a molded part. This can be especially true when molding something like a clarified polypropylene. Running cold molds with clarified polypropylene tends to minimize the size and growth of crystallinity and will produce a clearer part. But if the crystallinity of clarified polypropylene builds it is more likely to be cloudy.

For every material being molded the recommended process settings must be known. Verify the actual process results to the specified process settings. During process development try adjusting the mold temperature to determine impact on cloudiness issues.

21.3.2 Cloudiness Troubleshooting Mold Issues

Potential mold-related causes for cloudiness include:

- Water leak
- Surface of mold
- Hang up areas

21.3.2.1 Mold: Water Leak

If the mold has a crack that is allowing water to enter the cavity the result can be cloudiness. If cloudiness is the problem and other root causes have been eliminated inspect the mold for potential water leaks. A water leak in a mold can be difficult to detect. Sometimes water drops will be obvious on a mold surface but other times the drops will be very small and may not even appear until the mold has been pressurized with clamp tonnage. When checking for a cracked mold leaking water it is always a good practice to clamp up on the mold at tonnage to see if the clamp tonnage causes any water to seep through a crack.

21.3.2.2 Mold: Surface of Mold

To optimize the transparency of a molded part the mold surface should be polished. If the polish is inadequate light will be scattered resulting in a reduction in transparency. Areas of the mold that are not polished but rather are textured will also impact the scattering of light.

Another issue that can occur on the mold surface is buildup of material byproducts. The buildup can reduce the polish level on the mold resulting in a milky reduction in transparency.

Review the mold surface to make sure that there are no areas lacking in polish or have a surface buildup. If the mold surface is dull it may need to be cleaned with something like Zapox to remove buildup, or polished with diamond paste to correct an area lacking in polish.

21.3.2.3 Mold: Hang Up Areas

If the mold has a hot runner system there can be locations that trap material leading to drag out over time. Areas of particular concern include the sprue opening, any 90° corners, mismatch between components, valve pins, spreader tips, and internal heaters. Any area that results in a hang up of material can cause defects long after a material or color change has occurred. Even if only transparent material is ever run through the mold, a poor material change in the barrel can put contaminated material into the manifold.

If the mold will run different colors or materials ensure the hot runner supplier is informed prior to building the manifold. The supplier can focus on eliminating dead spots that can cause material to hang up.

21.3.3 Cloudiness Troubleshooting Machine Issues

The most likely cause of cloudiness relating to the molding machine would be hang up areas.

21.3.3.1 Machine: Hang Up Areas

If a molding machine has an area where material can be trapped and hang up the material change process will be more difficult. Hang up areas can be locations of mismatch between the barrel and end cap, end cap and nozzle, nozzle and nozzle tip, nozzle tip to sprue bushing, or damaged spots on the screw, barrel, end cap, or non-return valve.

Looking for hang up areas is a time-consuming process due to the requirement of disassembling the machine components. To justify the effort and time of looking for hang up areas it is critical to have exhausted other potential root causes. Sometimes the actual hang up can be a small step difference between components that allows wisps of the previous material to drag for extended periods of time. Figure 21.3 shows an off-the-shelf nozzle and nozzle tip with mismatch; the shiny inner diameter is a step difference between the nozzle body and the nozzle tip.

Figure 21.3 Hang up step (shiny inner diameter) looking through the end of a nozzle towards the tip

Sometimes if hang up areas are the root cause, it can be possible to help break the residue material free by raising temperatures and using a purging compound before running the new material through the barrel.

Something as simple as a nozzle tip can create hang up spots especially when using general-purpose tips, which have a dead spot that can trap material between color changes and will drag out as the machine runs. This is more critical on nozzle tips with small orifices because the dead spot is larger.

21.3.4 Cloudiness Troubleshooting Material Issues

Some possible causes of cloudiness in parts can be:

- Contamination
- Moisture content
- Semi-crystalline nucleation

21.3.4.1 Material: Contamination

One of the most common root causes for a cloudy appearance is a contaminated material. Contamination can come from a number of sources such as:

- Not keeping material containers covered, which can lead to various contaminants getting into a gaylord or drum. Any storage containers should have a lid to prevent contamination.
- Unclear labeling, which can lead to a person mistakenly adding the wrong material to a hopper or container. Labeling must be obvious to avoid people needing to try to figure out if they have the right material.
- Regrind, which can easily be mixed if it is not handled effectively. Immediate reuse at the point of scrap generation is ideal.
- Poor material changeover procedures, which can easily lead to a cross contamination of materials. Whether it is poor purging procedures or lack of thoroughness in cleaning hoppers the opportunity is there on every changeover.

Case Study: Material Mixing

In this case study a blend of crystal polystyrene and SBC (styrene -butadiene copolymer such as K resin®) was being used for a clear product. The crystal polystyrene was used to create a lower cost blend than running straight SBC. Even though both materials were naturally clear, creating a blend with too much polystyrene resulted in cloudy parts. The ratio of polystyrene to SBC had to be controlled to optimize cost, cosmetics, and performance.

21.3.4.2 Material: Moisture Content

High moisture content on materials can lead to excessive splay that results in a clear part looking cloudy. Verify the moisture content when processing hygroscopic materials. See Chapter 9 on drying for more information.

21.3.4.3 Material: Semi-Crystalline Nucleation

The way that semi-crystalline materials crystalize will impact how clear or opaque some materials end up being. A nucleated polypropylene can end up being fairly clear because the nucleation forms small crystals fast. If the nucleation level of the material varies the level of clarity will also vary. A polypropylene that does not contain nucleating agents will tend to be opaque and cloudy in normal appearance.

The nucleation effect with semi-crystalline materials will be impacted by wall stock, mold temperature, and melt temperature. Rapid cooling freezes the material in a more “amorphous” state and will maximize the clarity of the material.

22 Color Swirls

22.1 Description

Nonuniform distribution of color on a part will lead to a visual defect known as color swirls. Color swirls may appear as streaks, swirls, or patches of discolored material (see Figure 22.1).

Also known as: streaking, color streaks, marbling

Mistaken identities: black and brown streaking

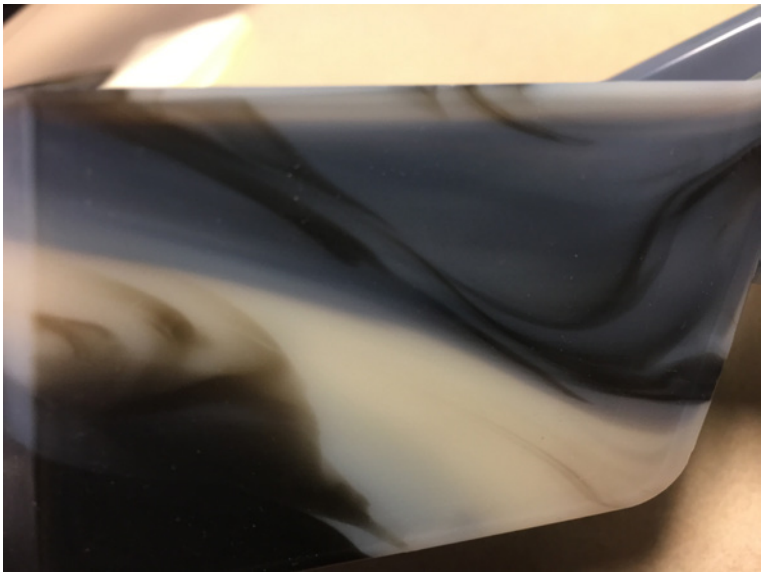


Figure 22.1 Color swirls

22.2 Color Swirls Troubleshooting Chart

Table 22.1 shows the color swirls troubleshooting chart.

Table 22.1 Color Swirls Troubleshooting Chart

Molding Process	Mold	Machine	Material
low back pressure		screw design	color concentrate
fast screw speed		inadequate back pressure	contamination
low melt temperature		hang up in screw or barrel	raw component issues

22.3 Color Swirls Troubleshooting

One of the first things to evaluate with color swirl problems is when the problem occurs. If the problem arises after changing from one color to another the solution will be different from random swirl problems or swirls that are always present.

If the color swirl problem is only occurring after changing material or colors it is time to focus on dedicated purging procedures. If changeovers are not done correctly it is possible that color swirls can continue for many hours.

As with all defects it is critical to ensure that the problem has been correctly identified. Sometimes color swirls are actually incorrectly identified black/brown streaks. Many of the fixes for color swirls will increase the severity of black/brown streaking so make sure that the problem is accurately identified.

22.3.1 Color Swirls Troubleshooting Molding Process Issues

There are a few basic process settings that will severely impact color mixing and swirling. As with troubleshooting any problem it is important to verify that the process settings match the documented process for the part being evaluated. Key items of focus for color swirls will be:

- Back pressure
- Screw speed
- Melt temperature

22.3.1.1 Molding Process: Back Pressure

Back pressure is a critical setting for applying shear to the plastic during screw recovery. The higher the back pressure is set the more mechanical work the screw will impart into the plastic during rotation. If back pressure is too low the melting material and color concentrate will not achieve adequate mixing, which will in turn lead to color swirls.

A common mistake relating to back pressure can arise during purging. Because the screw will often not recover with full back pressure process technicians will sometimes reduce or completely turn off back pressure when purging a machine, but if the back pressure is not returned to the set point the machine will often produce a number of defects, especially color swirls. Rather than dropping back pressure to purge, a way to combat this issue is to use a combination of screw rotate and decompress to build the shot to the required shot size. If the technicians consistently avoid dropping back pressure this failure mode will be eliminated.

When developing a process, it is critical to remember that higher back pressures will impart more mechanical energy and lead to a better mixing of material. If color mixing problems are being encountered higher back pressure will normally lead to significant improvements. Please remember though that back pressure is not a setting that should be maxed out for every process, as higher back pressure will potentially lead to overheated and degraded material as well as increased screw recovery time, which can cause cycle time increases.

22.3.1.2 Molding Process: Screw Recovery Speed

The longer the screw is allowed to rotate and mix material the better the color mix will be. If screw recovery is excessively fast, slowing down the recovery speed will typically provide a better mix. It is important to not slow down the screw speed to a point where the cycle time becomes delayed; the normal recommendation is that the screw should recover 2–3 seconds prior to mold open.

If a long cooling time is required for a part, screw rotate delay may be required to ensure that reasonable screw recovery settings can be used. If using a very slow recovery rate the material will not receive adequate shear energy to provide a uniform melt. For back pressure, like most settings on a molding machine, there is no generic set point that will work for every mold and process, so the proper set point must be established and maintained.

22.3.1.3 Molding Process: Melt Temperature

If material is not run at the proper melt temperature one potential outcome is poor mixing of color concentrate and nonuniform melt. Whether using a melt probe or an IR temperature gun it is critical to know the melt temperature that a process is providing. Melt temperature should be within the recommended range for the

material and should be documented for future process verification and troubleshooting. Always keep in mind that melt temperature is a result of many interacting factors including barrel temperature, screw design, back pressure, screw recovery rate, residence time, etc. Do not rely on barrel temperature settings as a predictor of melt temperature.

22.3.2 Color Swirls Troubleshooting Mold Issues

The mold is not typically the root cause of color swirls. The most likely tooling-related color swirl problem will be from color contamination from a hot runner system. This really is more of a contamination issue but highlights the importance of proper color changeover procedures. A hot runner system must be treated as part of the melt delivery system, which means that it must be cleaned out during color changes. A helpful hint when conducting a color change on a hot runner system is to raise the hot runner temperature settings by 30–50 °F to expedite the color change through the hot runner manifold and drops.

22.3.3 Color Swirls Troubleshooting Machine Issues

The molding machine itself can be the root cause of problems with color swirls. Some of the main concerns related to the molding machine include:

- Screw design
- Inadequate back pressure
- Hang up in screw or barrel

22.3.3.1 Machine: Screw Design

A general-purpose screw design is not an ideal screw for mixing of material and color concentrate. There are a wide variety of mixing style screws that will provide a more thorough mix as the machine melts and conveys the plastic. These screw designs have a variety of mixing elements incorporated to help create a more uniform melt. These “melt quality” screws will provide additional melt turbulence which helps distribute the color concentrate within the base resin.

Processors will often try to apply a “Band-Aid” to the problem of poor melt uniformity. These “Band-Aid” approaches include using mixing nozzles and mixing tips. These nozzles and tips have mixing elements that will provide additional shear and mixing as the material travels through these locations. These items can help with color concentrate mixing but will not be as effective as having a dedicated mixing screw. When using mixing nozzles and mixing tips the machine will experience additional pressure drop through the nozzle due to the restriction to flow.

This additional restriction can create a situation where the process is actually pressure limited due to the restriction. Another concern with mixing nozzles and mixing tips is the possibility that the mixing elements can trap material, potentially causing contamination and degradation of material.

Case Study: Screw Design

This case involved a PC/ABS automotive trim piece. The part was molded in a press with a general-purpose screw and was experiencing nearly 10% scrap due to color swirls. In an attempt to minimize the color swirl defects the back pressure of the machine was maximized, screw recovery was maximized, and a mixing nozzle was used, but all still with ongoing part defects. A new mixing screw was purchased for the machine and the mold was sampled. After the screw was replaced the color swirl problem on this mold was completely eliminated. Back pressure reduced from 300 psi hydraulic to 100 psi hydraulic and the mixing nozzle was eliminated.

22.3.3.2 Machine: Inadequate Back Pressure

It is crucial in processing to understand if the molding machine is actually doing what you are asking it to do. An all too common problem is a machine that is not actually achieving the set point for pressure. When considering back pressure, a low actual pressure on a molding machine can translate into poor color mixing and lead to color swirls. Actual pressure can be verified in a number of ways including on-press hydraulic gauges, secondary in-line gauges added to test ports, or process monitoring equipment such as the RJG eDART. If the machine has a back pressure setting of 150 psi hydraulic pressure but is only delivering 50 psi of actual pressure the machine must be checked for pressure calibration.

Also keep in mind the impact of the intensification ratio if moving a mold from one machine to another. You cannot expect the same melt quality if you use the same hydraulic pressure setting on a machine with a lower intensification ratio. The intensification ratio is a critical value to know for every machine to understand the actual pressure being applied to the plastic.

Example: Intensification Ratio

If machine “A” has a 15:1 intensification ratio a hydraulic back pressure set point of 100 psi will yield 1500 psi in actual plastic pressure. If this is compared to a machine that has a 9:1 intensification ratio the actual plastic pressure will only be 900 psi. This difference will potentially make a significant change to the quality of the melt produced by the machine.

22.3.3.3 Machine: Hang Up in Screw or Barrel

If there is damage to the screw, barrel, end cap, adapter, or nozzle, material can become trapped and not clean out well during color changes. These hang up areas can in turn lead to color swirls where the previous color drags out of the hang up location causing contamination and color swirls.

This is an area that can be checked after other key areas have been evaluated. Hang ups are more likely a root cause if the defect shows up after a color change and then gradually improves as the mold runs.

If hang ups are suspected it will be necessary to pull and examine the screw and barrel components. To help highlight hang up locations conduct a color change before pulling the screw and look for locations where the previous color is trapped.

22.3.4 Color Swirls Troubleshooting Material Issues

There can be a variety potential material issues that lead to color swirls including:

- Color concentrate
- Contamination
- Raw component issues

22.3.4.1 Material: Color Concentrate

The first potential root cause that can come from color concentrate is an incompatible material to concentrate problem (keep in mind that there are no "universal carriers") [1]. This can simply be an issue of using a concentrate that is intended for another material. If, for example, a styrenic concentrate is added to a PP base resin there may be color swirls due to poor intermixing between the materials. Therefore, the appropriate color concentrate should be used with a given base resin.

When using color concentrate the letdown ratio must match what is required for optimal coloring, for example 25:1 or 50:1. If the letdown percentage is low the color will not be dispersed adequately in the base resin. Also make sure that the correct size auger and settings are used with volumetric additive feeders. See Figure 22.2 for an example of a volumetric additive feeder. Different auger sizes will require adjustments to the feeder set point to achieve the correct letdown ratio.

Figure 22.2 Volumetric additive feeder

Be sure to understand the impact of different auger sizes. This is a volumetric relationship so the ratio in settings between a ½″ and a 1″ auger is not 2:1. To determine auger output consult the manual of the additive feeder for settings as well as minimum and maximum throughput rates.

Case Study: Color Concentrate

This case involved a PP automotive molding. The part ran in three different colors, black, gray, and tan, and only had issues when running with the tan color concentrate. Careful review of the data showed that this problem showed up virtually overnight and could generate over 20% scrap. Many things were tried to determine the cause of this issue including pulling and cleaning the screw and barrel. The color concentrate supplier was convinced that there was contamination coming from the barrel; however, running highly loaded white concentrate did not replicate the defect, and it would only show with the tan color concentrate. After convincing the color concentrate supplier that the defect was not caused by the machine (the mold also ran in a secondary press), the supplier conducted an in-depth investigation into their raw components. They determined that one of the raw pigments used in the tan concentrate was actually clumping together, which ▸

in turn led to areas of dark pigment that in turn became color streaks/swirls when processed. After the issue with the raw pigment had been corrected the problem was completely resolved. This was a case that took an extra amount of time to resolve because the color concentrate supplier had to be fully convinced that there was an issue with the material. Much waste could have been eliminated if the supplier had been more open-minded to the potential root cause.

22.3.4.2 Material: Contamination

Many common defects can occur when material is not protected from contamination (see Figure 22.3). There are many potential sources of contamination in a molding shop. Some general rules of thumb to help avoid contamination are:

- Keep material containers covered.
- Make sure that all hoppers, conveying lines, and feeders are thoroughly cleaned whenever materials are changed.
- Clearly label all materials to reduce the opportunity to mix materials. This can be everything from hoppers to gaylords to silos. Switching materials in a silo without a thorough clean out can lead to a long-term contamination issue that can lead to a variety of defects including color swirls.

Figure 22.3 Color swirl from contamination

Keep in mind that one pellet of contamination can lead to a color swirl in the molded part creating scrap. When troubleshooting a color swirl problem, it is well worth inspecting the material for contamination.

22.3.4.3 Material: Raw Component Issues

Occasionally there will be cases where a raw component of a material or concentrate will yield problems with distribution throughout the material. A potential issue can occur if the raw component does not disperse well with the color concentrate. Sometimes, raw pigments can agglomerate and clump together, which in turn can lead to areas of swirls.

This may especially be a factor when a mold has been running successfully and starts to suddenly experience color swirl issues when a new lot of color concentrate is introduced. If a problem starts to occur immediately after a new lot of material or color concentrate has been introduced then the material is a good place to start the troubleshooting process. There will be times when something in the material or the color concentrate is directly leading to the part defect.

One way to check for material issues is to switch to a different lot of material or even switch to another color material. If the issue is isolated to specific lots then it is critical to contact the material supplier to help with the investigation. Another important thing to keep in mind is that if the issue is in the raw material or color concentrate you will probably experience problems on any job that is running the material. Keep in mind that there are cases where certain molds will show a defect much faster and to a greater severity than others, so what may create 10% scrap on one mold may only cause 1% on another.

Case Study: Raw Component Issues

This example was using a concentrate for a polypropylene automotive component. In this case the new automotive interior specification called for a high loading on antistatic compound in the concentrate. It quickly became apparent that the 25 : 1 loading that was utilized on multiple parts was producing a wide range of final colors, varying in lightness (L value) as well as shifting red (a value), whereas the blue to yellow (b value) was ok. Visual confirmation of the L*a*b* color values showed that there was a definite shift in color on specific parts.

Root cause investigation was undertaken, which led to data that seemed to indicate the color was shifting based on barrel residence time. Further investigation with the material supplier determined that to meet the antistatic requirements they had to load the concentrate heavily with glycerol monostearate, which degraded with heat and residence time. The molds that ran with hotter melt temperatures and longer residence time delivered the worst results.

After many discussions regarding the specification and the concentrate impact it was determined that the specification was actually not the specific one that was dictated for this particular application. After the material supplier made some formulation adjustments the “new” formula was able to produce consistent and quality parts that met the customer’s need.

References

[1] Sepe, M., "Working with Color Concentrates", *Plastics Technology*, January 2012. https://www.ptonline.com/columns/working-with-color-concentrates

23 Contamination

23.1 Description

Contamination is a broad term that covers visual defects that appear in a molded part. Contamination may show up as specks of discoloration, streaks, splay, delamination, etc. See Figure 23.1 for an example of material contamination.

Also known as: black specks, black/brown streaks, color swirls

Mistaken identity: splay

Figure 23.1 Material contamination

23.2 Contamination Troubleshooting Chart

Table 23.1 shows the contamination troubleshooting chart.

Table 23.1 Contamination Troubleshooting Chart

Molding Process	Mold	Machine	Material
poor changeover	hot runner hang up	hang up areas	improper storage
high melt temperature	high hot runner temperature	anti-seize	regrind
	wear surfaces	robot contamination	incoming contamination
	lubricants		mixed materials
	cleaning		

23.3 Contamination Troubleshooting

There are a wide variety of ways that material can become contaminated. It can be a major challenge to try to work through all the possible routes of contamination. Most of the time the best place to start is with the material being brought to the molding machine and work backward from there.

23.3.1 Contamination Troubleshooting Molding Process Issues

Contamination can be brought on by several process issues including:

- Poor changeover
- High melt temperature

23.3.1.1 Molding Process: Poor Changeover

When a molding machine is stopped and goes through a material or color change there are many opportunities for contamination to occur. The entire feed system and melt delivery system must be thoroughly cleaned out to ensure that there is no residue of the prior material. Some of the key areas to examine include:

- Drying hopper. The drying hopper has several hang up areas including along the door edge, in the hopper loader, around the dispersion cone, the sample/drain tube, and in the distribution box. When conducting a material change all of these areas must be cleaned of all remains of the previous material. Hoppers can be

vacuumed out to help with clean out. Watch out for areas such as the sample tube because this is a spot that is often forgotten.

- Feed lines. Whether feeding the machine from the drying hopper, gaylord, or bag, it is critical to ensure that all of the material has been cleaned out. This is usually as simple as removing the feed line and allowing the vacuum loader to suck all of the material out of the feed lines. There have been cases were the material handler does a nice job of cleaning everything out but forgot the feed lines and when the new material was introduced all of the previous material was pulled in with it, contaminating the material.
- Machine hopper. The machine hopper will have a variety of locations where material can collect. Make sure to remove the magnet and clean out around the magnet drawer (see Figure 23.2). The hopper should be completely cleaned of all residue from the previous material. Also be sure to watch for any mismatch ledges between components such as the feed throat and hopper.
- Loaders. Loaders can also be a location that will trap material pellets. All loaders must be cleaned as part of the material changeover process.
- Additive feeder bins. Additive feeder bins must be cleaned when swapping colors. The feed screw on a volumetric feeder must also be removed and cleaned to avoid contamination.

Figure 23.2 Pellets stuck in magnet drawer

Often times molding shops will provide very little training for material handlers. If a material handler does not understand the importance of thorough cleaning during material changes they may cut corners which will lead to contamination. Provide a formal training for everyone in the shop that is responsible for loading materials to ensure that changeovers are executed correctly.

23.3.1.2 Molding Process: High Melt Temperature

When plastics are overheated they can degrade and contaminate the material with black specks or streaks. See Chapter 15 on black specks for more information about this problem.

23.3.2 Contamination Troubleshooting Mold Issues

Mold-related concerns that can cause contamination include:

- Hot runner hang up
- High hot runner temperatures
- Wear surfaces
- Lubricants
- Cleaning

23.3.2.1 Mold: Hot Runner Hang Up

Any areas in a hot runner system that can trap material can lead to contamination. Material trapped in hang up areas can continue to bleed into the melt stream long after the material change has been completed. When dealing with materials that are more temperature sensitive the trapped material may degrade and contaminate the parts with charred black specks. Figure 23.3 shows contamination from a hot drop tip.

Hot runner assemblies should be built in a way that at running temperatures there will be neither ledges nor mismatch locations that would hang up material. Drops must be securely assembled to the manifold so as to not leave any gaps between the drop and the manifold body.

Corners in hot runner manifolds can easily lead to dead spots where material hangs up. Depending on the hot runner design some manifolds will have corners that are gun drilled at 90° intersections. With a 90° intersection the outside of the corner will end up as a dead spot and material will be trapped in this location. Better designs have machined corner plugs with a curved corner, no dead spots.

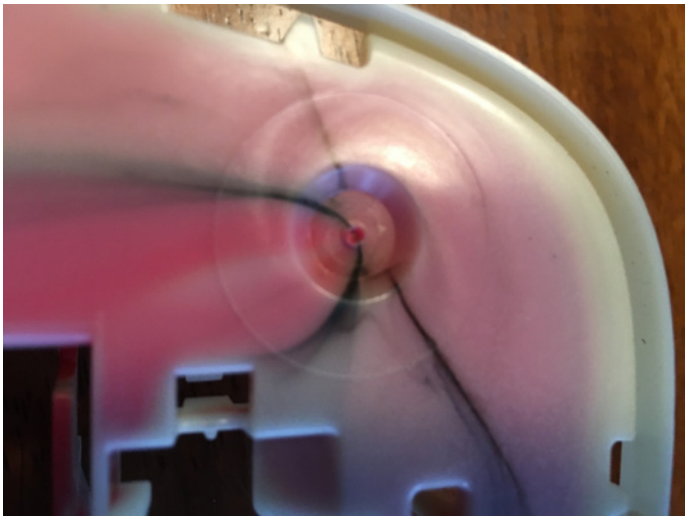


Figure 23.3 Contamination at hot tip

A manifold can be damaged and cracked which will provide a gap at the crack that will allow material to become trapped. Cracking a manifold is usually a result of a cold start where the manifold was not given enough time to heat soak. Cracked manifolds often need expensive and time-consuming repairs; therefore, educate process technicians to give a hot runner plenty of soak time before injecting plastic through the manifold.

23.3.2.2 Mold: High Hot Runner Temperatures

When hot runners are running at higher temperatures than is required for a material the odds of degrading material increase. As the material degrades in the hot runner system the result can be brown streaks or black specks contaminating the part.

Verify that the hot runner temperatures are set correctly and that actual temperatures are reading accurately. If a hot runner zone is constantly calling for heat it may indicate that a zone is either wired incorrectly or has a thermocouple misplaced.

23.3.2.3 Mold: Wear Surfaces

Any surfaces on a mold that rub against another surface have the opportunity to wear. Over time the metal dust or flakes that are worn from the surface can contaminate the cavity of the mold. This metallic dust will contaminate the molded part and may produce a visual defect.

Wear surfaces to keep an eye on include gibs, wear plates, die locks, cavity locking angles, and shutoffs. The dust that appears is often an early sign of galling starting so it is important to address any wear problems when they are detected.

23.3.2.4 Mold: Lubricants

Any of the lubricants used on a mold can lead to contamination of a molded part. Whether it is grease or oil, when excess lubricants reach the cavity they can contaminate the part producing scrap.

A key item to watch for is over-lubricating a mold. Frequently when a mold comes back from service, all moving components will have too much grease on them. There can be cases where a mold will bleed grease for hours of molding, producing nothing but scrap. Work with tooling sources to establish a standard method of lubricating a mold. It is not as simple as smearing some grease on the pins; grease should be lightly applied because over-greasing can lead to problems.

It is also important to find lubricants that work best for a given application. There is a wide variety of lubricants available and some perform better for specific applications.

23.3.2.5 Mold: Cleaning

Mold cleaning or lack of cleaning can both lead to contamination. For optimized processing a mold should be kept clean. Molds that are dirty can lead to surface contamination of molded parts. Caution must be used when cleaning molds, though, because mold wipes can leave behind fibers that can then be molded onto the surface of a part resulting in a squiggly, worm-like defect (see Figure 23.4).

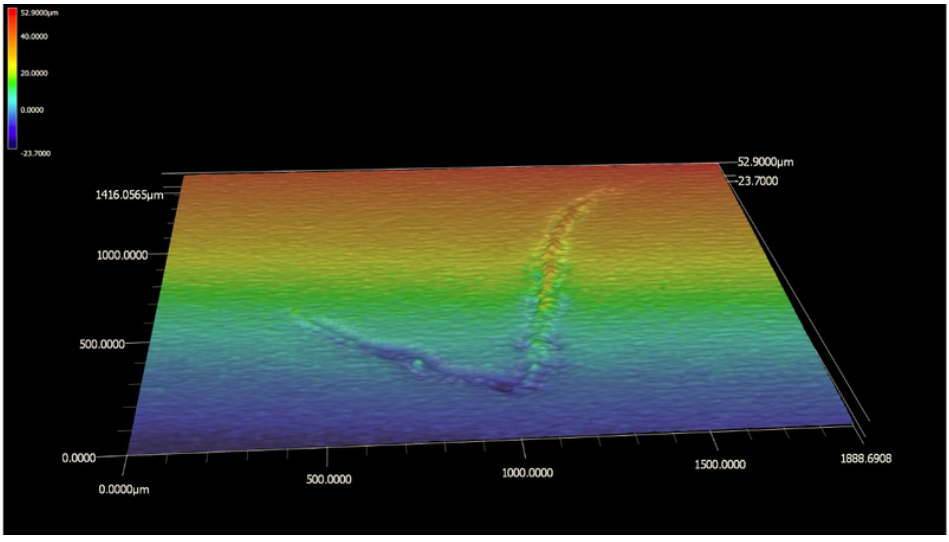


Figure 23.4 Surface scan of a mold wipe fiber molded onto a part surface

When wiping a mold surface, use clean lint-free wipes to minimize the opportunity for contaminating the mold surface. A dirty wipe not only leaves behind contaminants but may also scratch the mold surface.

Be aware that there are cases where the grease in the mold breaks down due to chemical contact. This can be magnified if someone sprays mold cleaner directly on the ejector half of the mold. The cleaner solvents can impact the viscosity of the grease leading to contamination of the molded part from the broken-down grease.

23.3.3 Contamination Troubleshooting Machine Issues

Some of the machine factors that can lead to contamination include:

- Hang up areas
- Anti-seize
- Robot contamination

23.3.3.1 Machine: Hang Up Areas

Any areas in the melt delivery system that have a mismatch can cause material to hang up. Some of the key areas that should be evaluated for hang ups include:

- Hopper to feed throat
- End cap to barrel
- Nozzle adapter to end cap
- Nozzle to nozzle adapter
- Nozzle tip to nozzle
- Nozzle tip to sprue bushing
- Damaged spots on screw, barrel, non-return valve, and end cap

It is a time-consuming process to disassemble these components and inspect for hang up areas but sometimes it is necessary to get to the root cause. Prior to shutting down for disassembly run another color of material through the barrel. This alternate color can help highlight spots where the original color is being trapped. Also look for areas of charred material when conducting this inspection. Again, this is a time-consuming process so make sure other potential causes are investigated first.

Examination of nozzle tips of different style will show locations where material can become trapped. See Figure 23.5 for cross sections of common nozzle tips: notice the potential dead spot at the ball end of the general-purpose style tip. General-purpose tips can cause contamination and streaking issues due to this dead spot.

Figure 23.5 Nozzle tips: general purpose, nylon, and full taper

23.3.3.2 Machine: Anti-seize

If people are conducting proper installation of new nozzles and nozzle tips they are applying thread anti-seize molded onto the nozzle threads to prior to assembly. If too much anti-seize is applied to the component it is possible to contaminate the plastic melt stream with the anti-seize (see Figure 23.6, for example). Anti-seize will usually result in dark streaks in the part.

Figure 23.6 Nozzle tip with excess anti-seize (top) and appropriate amount of anti-seize (bottom)

Anti-seize is an item where “if a little is good a lot is better” does not apply. Teach processors to use an appropriate amount of anti-seize to avoid running scrap at startup.

23.3.3.3 Machine: Robot Contamination

Take a look at the robots in the shop. Are they dusty with grease drips in various locations? When a robot enters the mold to pick a part it can contaminate the mold. One example of this is when a dirty air line brushes up against the cavity, because this will transfer some of the dirt and potentially lead to a cosmetic defect.

Another factor to consider with the robot is the condition of the end-of-arm tooling. If it is very dirty the vacuum cups can leave a surface contamination on the part. Also if components from the end of arm tooling touch the mold surface they may leave behind contamination that can end up in the cavity.

Case Study: Robot Contamination

This example was on a single-cavity ABS part that was experiencing what appeared to be splay. Examinations of drying conditions, moisture content, venting, process settings, and melt temperatures all led nowhere. Utilizing the STOP methodology and observing the process showed that when the robot picked the part it backed away from the ejector and a dirty air line brushed against the cavity. The cycle was stopped and a small smudge of grease/dirt was detected on the cavity. The part was molded and it was scrap due to the contamination. A couple of cable ties and some cleaning resolved the issue. In this case utilizing the STOP methodology earlier would have led to quicker effective troubleshooting.

23.3.4 Contamination Troubleshooting Material Issues

There are many ways that a material can become contaminated including:

- Improper storage
- Regrind
- Incoming contamination
- Mixed materials

23.3.4.1 Material: Improper Storage

In every molding shop there are a wide variety of foreign contaminants including:

- Dirt, dust, and pollen
- Other plastics

- Cardboard and wood
- Metal
- Grease and oils

If a material is left in an open container it is almost guaranteed that one or more of the above contaminants will make it into the material (see Figure 23.7). Any of the above contaminants will produce a contaminant in the molded part and cause a reject.

Figure 23.7 Open gaylord: contamination waiting to happen

It is very important to keep all material in covered containers to minimize the exposure to foreign materials. If a gaylord is sitting at a molding machine uncovered and someone blows out a hopper next to it, cross contamination will occur. Figure 23.8 shows contamination on the lid of a gaylord with wood chips which would have been in the material if the lid was not present.

Figure 23.8 Contamination on gaylord lid

23.3.4.2 Material: Regrind

Regrind provides another source of potential contamination. There are many ways that regrind can cause contamination including:

- Allowing parts from the wrong material to be ground with other materials. This can be avoided by either having people dedicated to grinding, or by only allowing parts to be ground at the machine where they were molded. Way too often is the case where an unaware operator throws a part into a grinder because the material is black.
- Regrind can become contaminated if it is stored before use. The same factors for keeping material in enclosed containers apply to regrind.
- When adding regrind to a process it is critical to make sure that everyone verifies that the regrind is added to the correct material.
- Contamination of the grinder itself whether from lack of cleaning or foreign material contamination.

It is always best if regrind can be reused at the point of generation. By adding the regrind back in at the molding machine many of the opportunities for contamination can be eliminated. Another advantage of point of use with regrind is that most times redrying of the material can be avoided.

Case Study: Regrind Contamination

This example was an ABS part being molded in a 700-ton molding machine. The machine was having problems with inconsistent shots that seemed to point to the non-return valve. When the front of the nozzle and end cap were ▸

removed, trapped metal was found in the nozzle and tip. Using the STOP process led to observation of parts being reground with metal inserts resulting in contaminated regrind.

Extra care must be taken if running regrind on parts that have inserts or assembly. Magnets will help catch ferrous metals but any non-ferrous metals such as brass will pass right through a magnet.

23.3.4.3 Material: Incoming Contamination

Every material has a level of "contamination" as produced. Most of the time the level of incoming material contamination is at levels that do not affect part quality.

For some fine-level molding applications materials are screened after manufacturing with a gel tape analysis (see Figure 23.9 for an example). Gel tape analysis can highlight if there are large or excessive contaminants in a material.

Figure 23.9 Gel tape

Contaminants that can be found in incoming material include:

- Rubber gels
- Agglomerates of ingredients
- Foreign material such as dirt, pollen, wood, or cardboard
- Carbon specks from degraded material occurring during the extrusion process

Case Study: Incoming Contaminants

In this case a PC/ABS was the material of choice. Parts had a high-gloss finish and were running without issues. A new lot of material was introduced and the parts started to show surface defects. Investigation with the material supplier showed that the TiO_2 used to color the material was agglomerating leading to visible surface defects during molding.

After a new lot of material was introduced the part defects were completely gone.

23.3.4.4 Materials: Mixed Materials

It is crucial that materials are kept from being mixed. If a foreign material is mixed with the desired material being molded, contamination defects are sure to occur. Some materials can even be dangerous when mixed, such as PVC and acetal. It is important to educate everyone in the plant that black plastic is not just black plastic and all materials must be kept separate.

Figure 23.10 shows an example of a contaminated part. The black TPO part was showing a small streak of gray on the surface. The part was cut through the streak and a distorted pellet was clearly visible in the part cross section. This is another example to show that one pellet of contamination is all it takes to make a defective part.

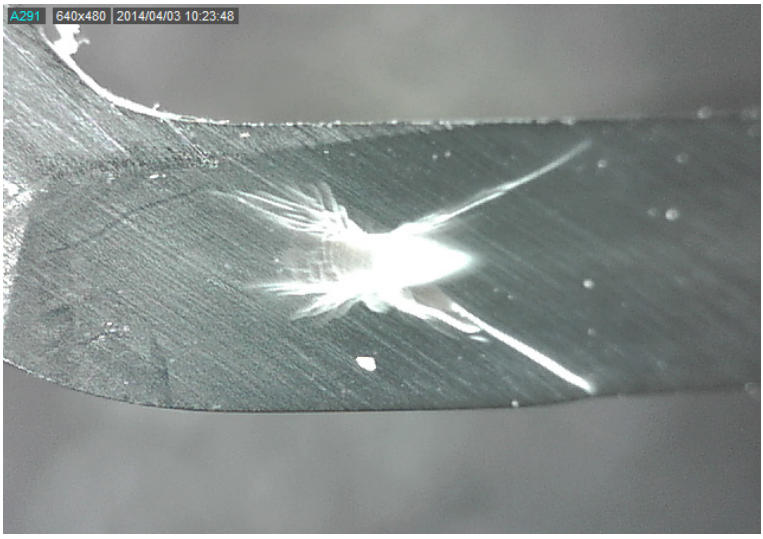


Figure 23.10 Single-pellet contaminant in wall stock

One key to avoiding material contamination is to clearly label all materials in every hopper or container. Before new material is loaded into a hopper or machine the label should be cross checked to verify that the correct material is being added. If a silo is contaminated the result can be thousands of pounds of scrapped material plus the lost time.

Complete cleanout of everything that a material touches is a must. All tankers, rail cars, feed lines, silos, hoppers, storage bins, and grinders must be cleaned thoroughly before introducing a new material. A few pellets left over during a changeover can yield scrap for hours. See Section 23.3.1.1 for more details on changeovers.

Mismatched materials can show as many different defects including contamination, delamination, black specks, color streaking, and splay.

24 Cracking

24.1 Description

Cracking is when the part has a physical fracture or break. See Figure 24.1 for an example.

Also known as: breaking, snapping, fracture

Mistaken identity: scratches

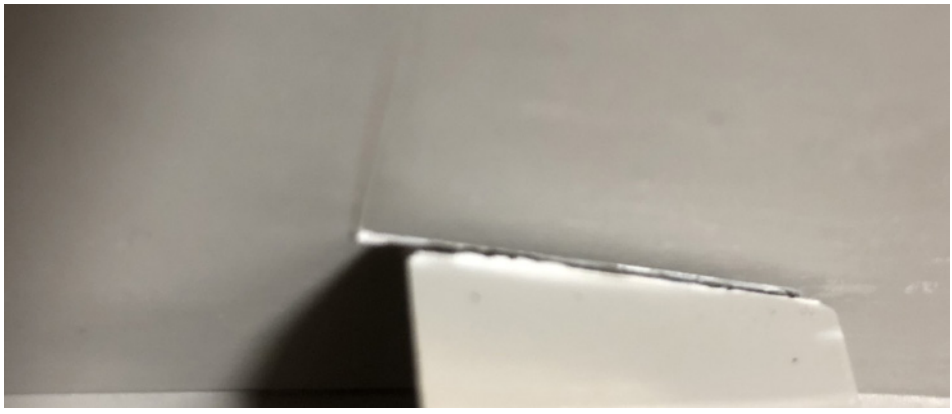


Figure 24.1 Crack at rib to wall intersection

24.2 Cracking Troubleshooting Chart

Table 24.1 shows the cracking troubleshooting chart.

Table 24.1 Cracking Troubleshooting Chart

Molding Process	Mold	Machine	Material
high second-stage pressure	undercuts	overheating	moisture content
melt temperature	sharp corners	lack of process control	contamination
residence time in barrel	lack of draft		regrind
ejection	lifter or slide corners		manufacturing issues
part removal			molecular weight
operator			wrong material
mold breakaway speed			
gate seal			

24.3 Cracking Troubleshooting

Cracking is a defect that can often come from post-molding conditions. When troubleshooting cracking it is important to understand when the cracking is occurring. Also with cracking, careful examination of the part design may highlight areas with sharp corners that are acting as stress risers and initiating the crack; therefore look for opportunities to add radius to these areas.

24.3.1 Cracking Troubleshooting Molding Process Issues

Cracking can be caused by several process settings as well as post-molding conditions. Some of the key factors to investigate for cracking include:

- Second-stage pressure
- Melt temperature
- Residence time in barrel
- Ejection
- Part removal
- Operator
- Post-mold operations

- Mold break away
- Gate seal

24.3.1.1 Molding Process: Second-Stage Pressure

If a part is over-packed it will stick more in the mold. This increased amount of sticking can lead to issues with parts that crack when the mold opens or during ejection. To check for sticking in the cover half of the mold, listen (STOP: Observation) for cracking and popping noises during mold open. Stop the mold before the part is ejected and examine the part as it sits on the ejector side of the mold because areas that stick in the cover will often be lifted away from the ejector. Note that over-packing is one potential cause for sticking that needs to be verified.

If second-stage pressure has been set high accidently, a simple process verification should catch the issue. If the mold is running in a different machine remember to account for intensification ratio differences.

During process development it is important to understand how much process window is available before the part starts to stick. This can be determined by conducting a simple experiment of adjusting second-stage pressure to determine at what pressure sticking becomes a problem. Remember to listen for noise during mold open and inspect the part on the ejector half of the mold to ensure it has not been pulled away from the core due to sticking in the cover.

24.3.1.2 Molding Process: Melt Temperature

Verification of melt temperature is a reasonable step when checking for cracking issues. If a material is being processed outside of its temperature range it will have the potential to degrade and lead to parts that are brittle enough to crack during ejection or subsequent part handling.

Bear in mind that higher melt temperatures could lead to over packing either throughout the part or in localized areas. This potential of over packing could lead to sticking that in turn could create cracks on the molded part.

24.3.1.3 Molding Process: Residence Time

Similar to high melt temperature, excessive residence time can lead to degradation of material, which in turn could create part cracking. Verification of shot size versus barrel capacity is important to ensure that the material is not spending excessive time at temperature. The typical suggestion is that the shot size to barrel ratio should be somewhere greater than 20%. If the shot to barrel percentage becomes too small then there will be an increased length of residence time. Materials will normally degrade thermally based on time at temperature, so the longer the residence time the less temperature the material can tolerate before degrading.

Residence time must be considered for hot runner manifolds as well as the barrel. The hot runner manifold is a continuation of the melt delivery stream and can impact the residence time of the material. Most manifolds are designed to clear out every shot but there can be cases of greater than one shot capacity in the manifold which adds to the overall residence time of the material at temperature.

Another factor impacting residence time is any hang up area in the system. If for example there is a step between the nozzle and the adapter there will be a dead spot that traps material. This trapped material can break free over time leading to degraded material being injected into the mold. This issue often will result in splay or black specks and streaking; however, it could also contribute to cracking in the molded part. To avoid this situation, it is important to match components on a molding machine and inspect for damage. Any known dead spots in material flow should be eliminated.

24.3.1.4 Molding Process: Ejection

Cracking will frequently occur during ejection. This is usually related to how hard the part is sticking in the core half of the mold. To identify if ejection is creating the crack on the part, check the part for cracks prior to ejecting (do this by stopping the molding cycle before ejecting). If cracks are not present then eject the part from the mold again looking for cracks. If the part appears to be cracking during ejection there is probably a root cause such as excessive shrink on the core or undercuts on the core. To try to eliminate cracking on ejection it is sometimes possible to adjust the ejection velocity to improve the problem. Please note that if the ejection velocity is slowed there will be a negative impact on cycle time.

Rather than modifying the ejection velocity a better solution is often to resolve the true cause. As an example, sometimes long cycle times will cause excessive shrink onto the core making ejection difficult, so speeding up the cycle can at times cure the problem. It is also important to review the mold for undercuts, EDM/machining marks, damage, and surface finish. Also verify that adequate ejection is provided during tooling design. Tooling concerns will be addressed in the tooling section below.

24.3.1.5 Molding Process: Part Removal

If cracking is not occurring during ejection it may be occurring after ejection. An example of this could be if the part is extracted by a robot and the robot may be damaging the part by hitting excessively hard or by making a move that damages the part. Another possible point of cracking a part is as the part hits the chute or conveyor if the part is not robotically picked. If the parts are dropping on a chute and cracking it is possible to cushion the chute with cardboard to prevent the damage. Some parts by design should be robotically handled to avoid damage from im-

pact onto a chute. If a part is robotically picked it can be damaged by the end-of-arm tooling as well as by being dropped from too high of a height. A key portion of STOP troubleshooting is Observation, which includes listening for strange noises.

24.3.1.6 Molding Process: Operator

Cracking is a defect that can be caused by how an operator is handling a part. If operators are dropping the part or banging it around they could actually cause the part to crack or break. Normally if an operator is causing the part to break the design has an inherent weakness that should be addressed. If a part is so weak that simply handling it causes damage it should be examined for improvement opportunities.

Also when dealing with operators, make sure that there are clear work instructions available and that they are being followed. It can be difficult to track a problem if every operator is treating the parts differently.

Another possible impact that an operator can have on parts is to chemically contaminate the molded part. An operator could try to improve the part by wiping with a cleaner. Various materials will react to solvents and cleaners and may show crazing or an environmental stress cracking from the chemical exposure. It is possible that someone trying to help could jeopardize the integrity of the molded part.

24.3.1.7 Molding Process: Mold Breakaway Speed

If a part has a tendency to stick in the cover half of the mold fast mold breakaway speeds can damage the part. There will be cases where running a slower breakaway speed will help to release the part from the cover. The downside of slowing down breakaway speed is that the cycle time will be increased. This can also be a factor when the crack is occurring near a slide driven by a horn pin.

Notice that the interaction with the mold must be carefully evaluated when dealing with breakaway problems. See tooling concerns below for details, and do not process around a tooling issue!

24.3.1.8 Molding Process: Gate Seal

There can be cases where molding a part with gate seal freezes in excess stress near the gate. Especially if the cracking is occurring near the gate it may be worth trying to mold some parts without gate seal to see if there is an over-packing high-stress condition contributing to the cracking. Try running without gate seal to see if the cracking problem is improved.

24.3.2 Cracking Troubleshooting Mold Issues

When evaluating the mold look for sharp spots that can create stress concentrations. Note that part designs should always accommodate and specify radius at corners and intersections. Typical mold concerns are:

- Undercuts
- Sharp corners
- Lack of draft
- Lifter or slide concerns

24.3.2.1 Mold: Undercuts

Undercuts in the mold steel will create localized spots where the plastic is "die locked" behind the steel. These undercuts will trap the part making it more difficult to remove from the mold (either cover or ejector side). The added sticking of the part may cause higher stress on the part during mold opening or ejection. Also undercuts may cause stress risers in the molded part.

Undercuts can be caused by several tooling-related issues including EDM or machining marks, burrs on the parting line, scratches, and damaged or broken steel. Close inspection of the mold steel for damage and the part for drags or damage may indicate areas of undercut plastic. These undercuts must be polished out of the mold.

24.3.2.2 Mold: Sharp Corners

Plastics do not like sharp corners, which concentrate stress where cracks will tend to emanate from. Any corners or intersections on a mold should have a radius rather than be produced in a "dead sharp" condition.

Small radii are hard to detect on mold parts. One quick and easy check is to take a ball point pen and draw along the radius of the molded part. If the part has no radius the ball point of the pen will bridge across the 90° and leave behind a double line, but if a radius is present the ball point will leave one line as it marks along the radius.

24.3.2.3 Mold: Lack of Draft

In cases where the draft angle is inadequate for ejection of the part the result is a tendency for the part to stick in the mold. If a part is sticking in the mold the proper draft angles and surface must be selected based on the specific material being molded.

Before a mold is built a thorough draft analysis must be conducted to ensure that the part will be moldable. To overcome a lack of draft it may be necessary to add a secondary coating to the mold surface to prevent the part from sticking.

24.3.2.4 Mold: Lifter or Slide Concerns

Cracking can occur if the part is sticking on mold details such as slides or lifters. Always verify that there is adequate draft on the mold component to allow release. In many cases the surface finish of the slide or lifter must be correct to allow release; this may mean a draw polish in many applications.

Check that the lifter or slide detail has enough travel to allow complete release of the part. There are cases where the slide or lifter does not have the required travel and causes the part to hang up on the detail which can lead to cracking or other damage to the molded part. This is another key item to verify during mold design reviews.

24.3.3 Cracking Troubleshooting Machine Issues

When it comes to cracking, the major machine issues will relate to the potential to degrade the material, including:

- Overheating
- Lack of control

24.3.3.1 Machine: Overheating

Anything that degrades a material can lead to cracking or breakage of molded parts. The degradation on the plastic will lead to a reduction in the material's physical properties, which can lead to this problem. Also, if material is degraded it may have a greater tendency to stick in the mold leading to higher force required for ejection and potentially causing damage to the part.

If all barrel zones are set to the correct temperature (per documented process) check that all zones are actually at the set point. Verify the percent amp draw on the zones, because if one zone is calling all of the time it may be overheating. A zone that calls for constant power may be reading an incorrect thermocouple. Usually if zone thermocouples are swapped one zone will be calling for power at all times but another zone will never call for power. When heat zone thermocouples are swapped there will be very inconsistent heat on the barrel. Newer controllers have thermocouple burnout detection that will help eliminate some issues with thermocouples.

24.3.3.2 Machine: Lack of Control

As mentioned above, lack of thermal control is a key potential to degrade material. In addition to the above points regarding temperature issues remember that shear input is the largest contributor to melt energy in processing plastic. If the back pressure or screw speed is not well controlled for the machine the excess shear can

degrade the plastic, which in turn can lead to problems with cracking or breaking. Verify that the actual back pressure and screw rotation speeds are equal to the required process values.

Another point of impact for machine control is second-stage pressure. If the second-stage pressure is not controlled the mold may be over-packed, which could lead to sticking. If a part sticks either during mold opening or during ejection, damage to the part may occur. Frequently when sticking is occurring cracking noises can be detected either at mold open or during ejection. Verification that the machine is providing an actual second-stage pressure that matches the required value is critical. To effectively troubleshoot it is critical to know what is actually occurring as the machine sequences through the process.

For further details see Chapter 8 on machine performance.

24.3.4 Cracking Troubleshooting Material Issues

Some potential material-related issues for cracking include:

- Moisture content
- Contamination
- Regrind
- Manufacturing issues
- Molecular weight
- Wrong material

24.3.4.1 Material: Moisture Content

Many materials will undergo hydrolysis when processed with excess moisture content. During hydrolysis the molecular chain length of the plastic will be broken down and shortened. As a result of this breakdown of the material the physical properties will experience a large drop off. This drop in physical properties will often lead to cracking of the part during ejection or post-mold handling. If cracking suddenly shows up verify that the material has been adequately dried.

As with many other defects it is critical to follow the recommended drying procedure for all materials. See Chapter 9 on drying for troubleshooting the drying process.

It is important to understand that not all materials will develop splay as soon as they are molded with excess moisture. Polyesters are a good example as they can be degraded to the point where the part will shatter if dropped after molding but they may not have splay. Sometimes the first indicator that the polyester is wet is when a part sticks and the ejector pins push right through the part.

Nylon is a material that has very different properties in a dry-as-molded (DAM) condition versus its normally moisture-stabilized condition. When molding nylon that has been dried extremely well there may be cases of parts acting brittle and cracking during ejection or handling. In most cases as long as the material was not damaged exposure to moisture will allow the material to achieve its conditioned physical properties. Do not over-dry nylon and ensure proper tooling and process conditions to minimize the impact of nylon in a dry-as-molded state.

See Chapter 9 on drying for more information

24.3.4.2 Material: Contamination

The most likely culprit for material-related cracking or breaking is contamination. Contamination of the intended raw material with an alternate material can yield a nonhomogeneous mixture that is severely compromised. Most of the time if contamination is the root cause there will be other visual defects on the part such as splay, delamination, streaking, etc.

It is crucial to avoid situations that can cross contaminate materials. Preventative measures include keeping all material in closed containers, ensuring adequate cleanout of any storage bins, hoppers, or silos, clear labeling standards, education of employees on handling regrind, education of employees that the same color does not mean the same material, etc.

Make sure that when using color concentrates that the concentrate is intended for use in the base resin. Also maintain clear labeling and protocol to ensure that no-one grabs just any box of concentrate that appears to look like the right color. Not all concentrates are compatible with all base materials. In STOP troubleshooting it is critical to Think about possible sources of contamination.

24.3.4.3 Material: Regrind

If regrind is handled well it can perform as well as virgin material. However, regrind can create issues if the following occurs:

- Contamination with other materials
- Not drying if the regrind becomes wet
- Heat histories can lead to components in the additive package burning after reuse

24.3.4.4 Material: Manufacturing Issues

Material suppliers typically do an excellent job of manufacturing the materials that they produce. However, like any manufacturing process there will be variation and mistakes. If a component is left out of the formulation or not added at the correct level the material could create issues. Two possible culprits could be impact

modifiers and release agents. Impact modifiers missing from the formula could lead to a material that cracks when ejected from the mold and missing release agents will lead to sticking in the mold. These are rare cases because outgoing inspections on material properties should catch any problems if they slip through the production monitoring side of things.

24.3.4.5 Material: Molecular Weight

Material is produced in a manufacturing process and as such will have variation. One of the key items molders must deal with is variation in the average molecular weight. If the average molecular weight of a material is low the material will tend to get weaker from a physical property standpoint, which could lead to cracking of the part. Some mold and material combinations will be prone to this but it will be rare. Spending a lot of effort investigating average molecular weight will be very costly.

Unfortunately, average molecular weight is not easy to measure and keep track of in normal molding shops. The test that gets used to try to simply quantify a material is the melt flow index (MFI). If the average molecular weight is varying, MFI may detect the change.

24.3.4.6 Material: Wrong Material

Always verify that the correct material is being used. It is important to follow all documented standards when handling materials to ensure that materials are not used accidently.

Also bear in mind when looking at alternate materials that just because the specifications look good on paper does not mean that the materials will yield the same molding results. If an alternate material is being reviewed anticipate that process adjustments may be required. This is another point of STOP troubleshooting where Observation can help resolve issues quickly.

25 Delamination

25.1 Description

Delamination is a defect where layers of plastic separate from each other. This stratification of plastic will lead to both cosmetic and physical property concerns. A typical delamination of incompatible materials is shown in Figure 25.1.

Also known as: contamination

Mistaken identities: splay, silver streaking

Figure 25.1 Typical delamination of incompatible materials

25.2 Delamination Troubleshooting Chart

Table 25.1 shows the delamination troubleshooting chart.

Table 25.1 Delamination Troubleshooting Chart

Molding Process	Mold	Machine	Material
melt temperature	hang up areas	machine performance	contamination
fill velocity	lubricant contamination	hang up areas	incompatible color concentrate
residence time in barrel			

25.3 Delamination Troubleshooting

The first thing to look at in the case of delamination problems is the material. Many cases of delamination end up going back to a material contamination problem.

25.3.1 Delamination Troubleshooting Molding Process Issues

Process settings that may influence delamination could include:

- Melt temperature
- Fill velocity
- Residence time in barrel

25.3.1.1 Molding Process: Melt Temperature

Verification of melt temperature can expose many defects. There are cases where, if the melt temperature of a material is too high, the degradation that occurs can lead to layers delaminating in the material. If the melt temperature is the root cause there will also likely be other visual defects on the part including splay, black/brown streaking, and black specks. If the melt temperature is outside of the recommended range for the material and it is the suspected root cause then changes should be made to either the barrel temperatures, back pressure, or screw recovery speed.

25.3.1.2 Molding Process: Fill Velocity

On rare occasions excessive shear at a gate can leave a delamination in the material near the gate. This is more likely to occur on a material blend such as a PC/ABS material. Slowing down the fill speed will decrease the shear through the gate and may eliminate the delamination.

Sometimes with sub gates or cashew gates there will be delamination showing where the gate separates from the part. This can sometimes be fixed with a change in velocity through this portion of the process.

25.3.1.3 Molding Process: Residence Time in Barrel

Much like excess melt temperature, excessive residence time can lead to degradation of the material, which in turn can produce areas of separation or delamination on the skin of the part. Depending on who you ask, the recommended barrel utilization number may vary but it will usually be a range of approximately 25–75% utilized per shot. If the shot turnover rate through the barrel is low (under 25%) there will be many shots sitting at temperature. This becomes more of an issue as the cycle time lengthens because the time at temperature will grow longer with the increased cycle time. The more temperature sensitive a material is, the easier it will degrade with excessive residence time.

If a mold has to run in a press with poor shot turnover, one of the best things that can be done would be to profile the barrel temperature settings (low in the rear) and try to keep the melt temperature at the low end of the recommended specification. These changes will at least minimize how much heat the material is exposed to as it travels through the barrel.

With materials that are highly temperature sensitive such as PVC or acetal the residence time in the barrel is even more critical to ensure that the material is not degraded. One goal of molding is to try to deliver a plastic melt that will provide as much retention of the physical properties of the material as is possible.

25.3.2 Delamination Troubleshooting Mold Issues

There are a couple of basic things to look for as contributors from the mold for delamination:

- Hang up spots
- Lubricant contamination

25.3.2.1 Mold: Hang Up Spots

If a hot runner system has spots where the material can hang up and become “dead” the material may degrade over time and lead to delamination. When mate-

rial is degraded it can show a number of defects including delamination, splay, black specks, and brown streaking.

Also, if there is a mismatch between the sprue bushing and the nozzle orifice, material may become trapped, hang up, and degrade. For cold sprues nozzle tips should be 1/32 in under-sized to the sprue orifice size, and if running a hot runner the tip and sprue orifice should be 1 to 1 in size. Dead spots between components must be avoided to eliminate any potential material hang up areas.

25.3.2.2 Mold: Lubricant Contamination

If grease or oil from a mold is allowed to get into the cavity the result could be a delamination-type appearance. The lubricant will contaminate the material and potentially leave a blistered delaminating surface where the contamination is in the plastic.

Watch for hydraulic cylinders that are leaking as they can leak into the cavity. Also beware that over-lubricating mold components can result in contamination of the cavity with the lubricant. Many times when a mold comes back from being serviced the tool shop will have taken the approach of "if a little grease is good a lot is better", and this can lead to hours of defective product being run due to the contamination of the lubricant.

25.3.3 Delamination Troubleshooting Machine Issues

The machine itself can lead to delamination problems; the most common reasons are:

- Machine performance
- Hang up areas

25.3.3.1 Machine: Machine Performance

See Chapter 8 on machine performance.

25.3.3.2 Machine: Hang Up Areas

The main potential machine cause for delamination is due to areas of hang up such as between the end cap and the nozzle. If there are any steps in the steel as one component transitions to the next one there will be risk of trapping material. When material is trapped in the barrel or barrel components it will be dragged into the melt over time. This could lead to contamination with the previous material, which could be incompatible with the current material. Another effect of trapping material can be degradation of the material, which in turn can lead to delamination-type defects on the part surface.

Keep in mind that even though the throughput through the barrel is good, hang up areas will change the game for residence time.

25.3.4 Delamination Troubleshooting Material Issues

The reasons for delamination defects attributed to material issues usually fall into a couple of main categories:

- Contamination of material
- Incompatible color concentrate

25.3.4.1 Material: Contamination

If a material is contaminated with a foreign material there is a good chance that delamination will occur. This will obviously be more prevalent if the contaminating material is incompatible with the base resin; for example, if ABS and polypropylene are mixed you will have a problem, but on the other hand mixing polypropylene and polyethylene is not problematic.

It is key during all material handling to ensure that materials are kept clean and separated. There are many ways that materials can be contaminated including:

- Uncovered containers. If a box, barrel, bin, hopper, or gaylord is left uncovered there is an opportunity for foreign debris or material to get into the container. Something as simple as someone using an air hose to blow out a hopper on the next machine could lead to contamination in uncovered containers.
- Lack of verification. Sometimes people will dump a material into another container based on the color or appearance of the pellets without verifying that the materials are the same. This can also be an issue if someone switches over a feed line without verifying that the correct material is being sent to the correct location. It is therefore critical to clearly label all materials including regrind. Make sure that anyone that handles material understands the importance of getting the correct material into the correct containers. Also make sure that everyone who grinds runners or parts understands the correct grinder to use. Also when a silo is to be loaded make sure that someone has verified that the tanker is hooking up to the correct silo (this can be set up with different fittings as a poka-yoke).

Case Study: Material Contamination Causing Delamination

This particular case was at a molding shop that used mostly precolored material. There was one job that ran with natural base resin and a color concentrate. During an off shift the color concentrate ran out so the material handler went looking for more concentrate. They looked high and low and ▸

finally found what they believed to be more concentrate on a storage rack. They loaded up the “new concentrate” and started running parts. It was later discovered that they had grabbed an alternate material in black, not a concentrate, and were running contaminated parts. Close inspection of the parts showed an increase in scrap from splay and delamination because the materials were incompatible. All of the parts run during the shift were scrapped out due to the contamination. If someone had not assumed that the black material was OK to use, this problem would not have occurred.

25.3.4.2 Material: Incompatible Color Concentrate

When a concentrate is used to color a natural base resin, be sure to use a concentrate that matches up with the base resin. Although there are concentrates that can be used across different materials it is critical to make sure that the concentrate is intended for the desired material. Communication with the concentrate supplier will help resolve potential issues with using the wrong color concentrate. Also be sure to clearly document the bill of materials for a part so that everyone can quickly see what the base material is as well as the specific color concentrate to be used.

If the concentrate is suspected as the culprit, verify that someone did not grab a similar concentrate by accident (again make sure to clearly label and document). Also, a call to the concentrate supplier will verify that a particular concentrate number is intended for the base material being molded.

26 Dimensions

26.1 Description

One of the biggest ongoing potential problems in molding is dimensional control. Plastic will tend to grow and shrink based on environmental conditions. Variations during the processing of plastic will also lead to dimensional variation. Any given part may end up too large or too small or even have one dimension that is acceptable and another that is unacceptable.

Also known as: too big, too small, inconsistent, shrink

Mistaken identity: warp

26.2 Dimensions Troubleshooting Chart

Table 26.1 shows the dimensions troubleshooting chart.

Table 26.1 Dimensions Troubleshooting Chart

Molding Process	Mold	Machine	Material
second-stage pressure	gate size	nozzle tip	filler
second-stage time	sprue and runner size	velocity to pressure transfer	nucleating agents
fill velocity	cooling	non-return valve	type
fill only weight	hot runner control	machine performance	
melt temperature	component deflection		
mold temperature			
cooling time			
decompression			
cushion			

26.3 Dimensions Troubleshooting

Most factors that impact dimensions will be capable of either reducing or increasing a part size depending on which direction the change is made.

26.3.1 Dimensions Troubleshooting Molding Process Issues

There are many process factors that will impact dimensions on a part including:

- Second-stage pressure
- Second-stage time
- Fill velocity
- Fill only condition
- Melt temperature
- Mold temperature
- Cooling time
- Decompression
- Cushion

26.3.1.1 Molding Process: Second-Stage Pressure

One of the biggest impacts on dimensions will come from adjusting second-stage pressure. Normally higher pressure will pack more plastic in the mold leading to a larger part. Lower second-stage pressure normally leads to a smaller part.

With pressure it is important to realize that once the gate or wall stock freezes there will be no more pressure transferred through the part. This is an important aspect of maintaining uniform nominal wall stock. If a part has varying wall stock the thin areas will tend to freeze, which leads to additional shrink in the thicker areas, which in turn may limit impact on dimensions. Once the gate or wall freezes there is no longer a chance to pack the part. If the area of concern is a thin wall section the wall stock may freeze so fast that there will be very little change in dimensions.

Although keeping things simple is always the preferred choice there will be cases that using a profiled hold may be required to impact a challenging part. Sometimes using a high second-stage pressure for a couple of seconds will help pack out a thin section. It is important to understand, however, that adding a profiled second-stage pressure increases the complexity of the process and creates another thing that needs to be validated when running parts.

It is key to understand that the actual pressure applied in the cavity is what drives dimensions, not the set point on the molding machine. In many cases the best available data a processor has to work with is the set pressure of the machine, but an improved tool that gives a much better understanding of what is occurring inside the cavity is a cavity pressure transducer (see Figure 26.1).

Figure 26.1 RJG cavity pressure transducer

26.3.1.2 Molding Process: Second-Stage Time

It is critical to determine if a mold is going to run with or without gate seal. If gate seal is required it is critical to have enough time on second stage to ensure that gate seal will be consistent. If a mold is running on the edge of gate seal, variation will be noted in the quality of the parts especially dimensionally.

Always conduct a gate seal study during process development and establish whether the mold will run with or without gate seal. There are different ways to establish what second-stage time should be set at: some people will add 1 second to the gate seal time or add 10% to the gate seal time. Determine a method and make sure that it is followed. If the gate seals at 8 seconds, setting second-stage time at 8 may lead to times when the gate is sealed or unsealed; therefore, setting second-stage time at 9 seconds will make sure that the gate is sealed. See Figure 26.2 for an example of a gate seal study form; note in this example that the gate seals at

8 seconds, so second-stage time would be set to 9. Any time longer than 9 seconds for second stage will only be packing the runner because if the gate is sealed the pressure can no longer have any effect past the gate. Molders do not get paid to sell runners, so try not waste time and material packing out runners!

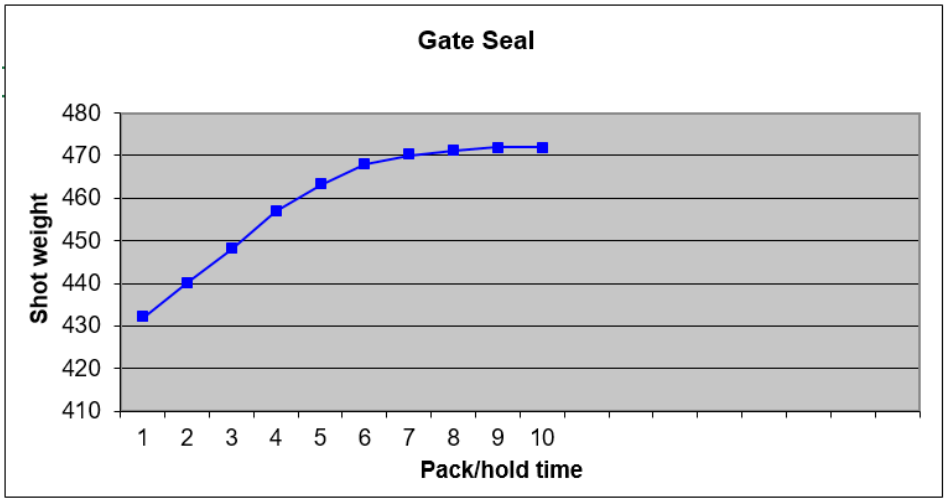


Figure 26.2 Gate seal study form

Another key point is that once the gate is sealed additional second-stage time will not pack more plastic into the cavity. The extra time may impact dimensions but only in the same fashion as what increased cooling time would do. Extra second-stage time beyond what is required for gate seal (plus safety factor) is only wasting energy as the machine packs out the sprue and runner. If you find that increasing second-stage time past gate seal is impacting dimensions try to replace the extra time from second stage and add it to cooling time.

Remember that gate seal time can interact with the process, and increases in melt temperature, mold temperature, fill speeds, and pack pressures may increase the gate seal time.

26.3.1.3 Molding Process: Fill Velocity

Changes in fill velocity will impact the viscosity of the material being molded. At slow fill rates the frozen wall will become thicker during fill and will limit the ability to effectively pack out the part. If the part is not packed out well the dimensions may vary or not meet requirements. Volumetric flow rate needs to be maintained during molding runs as well as from run to run. Consistent mold fill will lead to consistent packing of the part and consistent dimensions.

26.3.1.4 Molding Process: Fill Only Condition

When running with a Decoupled™ process the amount of plastic injected during first stage will impact how the part is ultimately packed out. Second-stage pressure should be removed from the process to verify that the fill only weight is accurate to the required 95–98% specified on the setup sheet. Part of STOP troubleshooting is to Systematically work through a problem and running a fill only shot is a good place to start.

If the fill only shot is light and a lack of plastic was injected during first stage, the resulting cavity pressure will tend to be lower, which will in turn lead to a smaller part. If the fill only shot is light then the transfer position should be adjusted to provide a fill only shot that matches the specified weight.

If the fill only shot is heavy there is excessive plastic being injected during first stage. If the shot is transferred beyond 99% fill only weight then the part is actually seeing pack during fill, which can lead to increased part size. Again verify that the fill only shot matches the specified value and is approximately 95–98% full.

26.3.1.5 Molding Process: Melt Temperature

Melt temperature is a factor that can impact shrink in both directions.

A higher melt temperature will decrease the pressure differential from the gate across the cavity thereby increasing the ability of the part to be packed out. On the other hand, a hotter melt will lead to higher shrinkage due to cooling, especially by post-mold shrinkage.

On a process that has been running well it is important to verify that the melt temperature matches the documented process. If the baseline melt temperature is unknown or the process has never produced dimensionally acceptable parts, then it may be necessary to adjust the melt temperature to determine the impact on the dimensions. Remember that melt temperature changes can make the dimensions go either direction.

Also increased melt temperature may be significant enough to impact gate seal time. Verify that the gate seal time is adequate when changing melt temperature, fill speed, mold temperatures, or second-stage pressure.

Figure 26.3 shows data from an RJG eDART process monitoring system that shows one cycle that had an extended rotate time due to a material feed issue. The following shot had an increase in cavity pressure that led to a dimensionally inaccurate part.

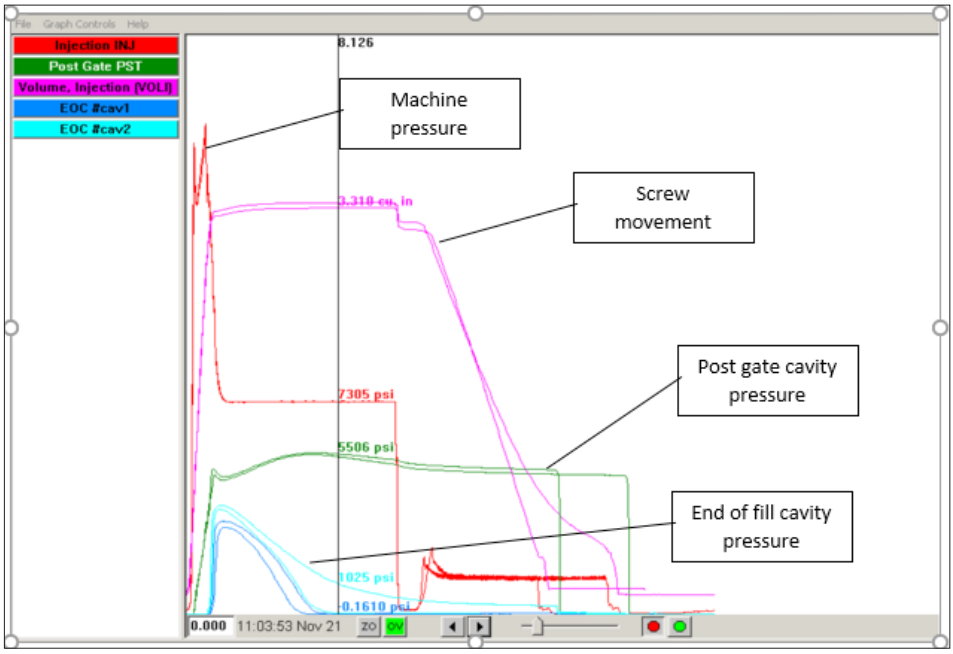


Figure 26.3 RJG eDART® cycle graph: the slight change in recovery time led to a part that was dimensionally out of specification on the following shot

26.3.1.6 Molding Process: Mold Temperature

Normally an increase in mold temperature will lead to increased shrink in a part. This increase in shrink will provide a smaller part. Much of this increased shrink will come as post-mold shrink. If parts are too large an increase in mold temperature may help, but if they are too small decreasing mold temperature may help lock in shrink.

Care should be taken with mold temperatures when molding semi-crystalline materials because in this case a cold mold may lock in shrink to a point, but exposure to higher post mold temperatures may cause the part to shrink and warp to a lower stress state.

Many factors need to be evaluated for optimal mold cooling (see Chapter 14 on optimized mold cooling for details).

26.3.1.7 Molding Process: Cooling Time

Increasing cooling time and thereby cycle time usually will help make a part larger. This is however an expensive way to increase dimensions because your machine and mold have now become an expensive cooling fixture.

Due to the cost of cycle time increases, adding cooling time is typically an unattractive option for changing dimensions. If cooling time improves the dimensions it would be well worth investigating ways to improve the cooling of the mold to minimize the cycle time.

26.3.1.8 Molding Process: Decompression

If a process is running without adequate decompression the non-return valve will not set consistently, which in turn can lead to issues with dimensions. Verify that the machine is running with decompression and that the cushion is not showing large variations and/or bottoming out. Adding a small amount to decompression and verifying impact is a simple item to validate during troubleshooting. Different non-return valves will react differently to decompression and some experimentation may be required.

26.3.1.9 Molding Process: Cushion

Normally the key to watch for on cushion is verifying that the machine is running with a cushion. If the screw bottoms out, the machine will not be able to transfer pressure through the mold, which will lead to a part that would be small dimensionally.

Lack of cushion could be from the shot size being set too small, plastic leaking at the nozzle or hot runner, lack of decompression, or a machine issue (see below). Before adding material to the shot size verify where the plastic is going. Automatically adding shot when the press is leaking at the nozzle will create damage to the heaters and thermocouples on the barrel as the plastic starts to encapsulate the barrel. If the press has been running fine, first verify where the plastic is going before adding shot!

26.3.2 Dimensions Troubleshooting Mold Issues

Possible mold related concerns are:

- Gate size
- Sprue and runner size
- Cooling
- Hot runner control
- Component deflection

26.3.2.1 Mold: Gate Size and Location

If the gate size of a molded part is too small the part will not be effectively packed and may experience excessive shrink. If a gate is small it can freeze prematurely,

which impacts how well the second-stage pressure is applied through the cavity. Note also that oversized gates can lead to over-pack conditions that can lead to a part that is too large near the gate.

Flow analysis software can be used to determine the required fill pressure through a given gate size, which will help minimize the risk of a process being pressure limited due to gating. Flow analysis software will also provide a frozen layer plot that will indicate if the wall stock is still shrinking after the gate has frozen.

Also make sure to account for gate spacing. If the flow distance from a gate is excessive the part will have uneven distribution of pack pressure through the cavity. Plastic can only flow and pack at limited distances from a gate; this is influenced by factors including wall thickness, material viscosity, melt temperature, and fill velocity. Plan ahead to provide an adequate number of gates to effectively fill and pack a mold. Total volume of gate area should be considered for the amount of plastic to fill the mold; often times a wider but thinner gate will help fill a mold and still allow adequate pressurization of the cavity.

When gating it is important to ensure that adequate pack pressure can be distributed within the cavity. An area of concern to watch for is thin to thick transitions in the wall stock. If a thick section of the part is located downstream of a thin section the thin wall stock will freeze and limit the ability to pack the thick sections.

26.3.2.2 Mold: Sprue and Runner Size

Restrictions in the sprue, runner, or hot runner system can lead to pressure drops that limit the ability to pressurize the cavity. A sprue and runner must be sized to provide adequate pressurization to the cavity without being oversized, which will increase cycle time. STOP troubleshooting questions every step of the material flow path from the end of the cavity all the way back through the nozzle of the machine.

Be aware that too large sprue and runner sizes can contribute to situations where a part may be over-packed. Do not assume that larger gates, runners, and sprues are always a good thing, because they impact cycle time and may lead to cases of over-packing that can influence dimensions and warp.

26.3.2.3 Mold: Cooling

Mold temperatures that are not running at the desired point can result in differences in cooling. If the mold is running cool it may freeze the part more quickly resulting in a smaller part. A hot mold may allow the part to pack out better resulting in reduced shrink but also may impact warp and crystallinity, which could lead to increased shrink. Verify uniform cooling on the mold surface and evaluate the temperature versus the recommended mold temperature for the material.

Differential cooling in a part can result in differential shrink through a part. Differential cooling is also a major factor influencing warp and warp can impact the observed dimensions on a molded part.

Make sure that cavity details and coring have adequate cooling to provide uniform surface temperatures. There are cases where cycle times are dictated due to a specific location on a mold with poor cooling that in turn impacts the overall dimensions of the part.

See Chapter 14 on cooling for more in-depth discussion of cooling.

26.3.2.4 Mold: Hot Runner Control

The hot runner controller on a mold must provide reliable and repeatable temperatures for all of the hot runner zones. If a hot runner zone is not running to the desired set point it can impact the ability to fill and pack the cavity.

Verify that valve gates are actuating at the desired point of the process. If a valve gate fails to open, the mold will fill and pack differently from the original established process. Valve gates need to be serviced over time and the cylinders may need to be rebuilt to provide proper valve actuation.

26.3.2.5 Mold: Component Deflection

In some cases components of the mold or the mold itself may have deflection that in turn leads to a dimensional issue on a specific detail of the mold. Deflection of a lifter could lead to a dimension on a snap detail that is different to that intended in the design. Thickness of part details can change if the mold or clamp is deflecting.

Case Study: Clamp Deflection

In this case an acetal part was being molded in a 170-ton press. The overall part was having consistency problems with thickness, which led to stack up problems. Examination of the part thickness versus the cavity dimensions showed that the molded part was actually sometimes thicker than the cavity was cut to. The mold was moved into a 230-ton press and the dimensional issue was resolved. The mold was actually being blown open and the part was thicker but did not show flash.

26.3.3 Dimensions Troubleshooting Machine Issues

Potential machine issues include:

- Nozzle tip
- Velocity to pressure transfer

- Non-return valve
- Machine performance

26.3.3.1 Machine: Nozzle Tip

When troubleshooting dimensional issues it is important to verify that the correct pressure is being supplied to the cavity. One often-overlooked problem area is the nozzle tip. If the wrong tip is being used the pressure drop from the machine will be higher. Care must be taken to ensure that the correct orifice is used as well as the correct style (i.e., nylon versus general purpose). Nozzle tip orifice and style must be documented and matched every time with the mold. Figure 26.4 shows a cavity pressure comparison between the same process running with a general-purpose 3/16-in tip and a general-purpose 5/32-in tip. On this process a one-size change in tip orifice had a huge impact on the plastic conditions. Tip orifice cannot be taken for granted, ever!

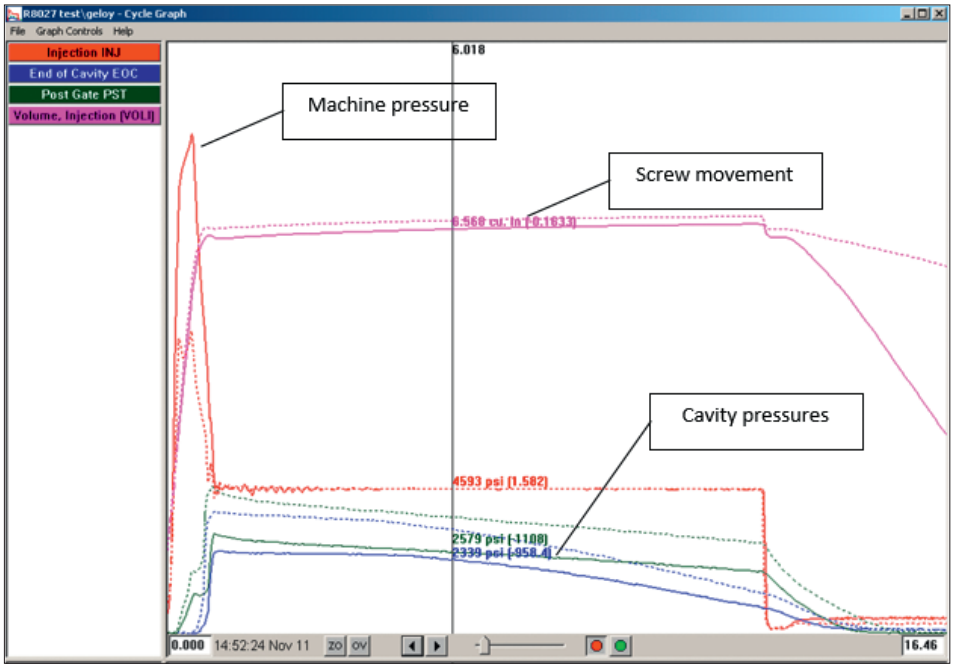


Figure 26.4 RJG eDART® cycle graph showing variation in cavity pressure between cycles run with a 3/16-in versus a 5/32-in nozzle tip

One method for ensuring that the correct nozzle tip is always used is to weld a 7/8-in nut on the side of the mold and then store the tip for that mold threaded into the nut. Every time a mold change is performed the nozzle tip is swapped for the one stored on the mold.

26.3.3.2 Machine: Velocity to Pressure Transfer

When the machine hits transfer it switches over from controlling velocity to controlling pressure, and this critical part of the process is often overlooked. Figure 26.5 shows a poor velocity to pressure transfer. Note the large dip in machine pressure at transfer as well as the cavity pressure dip.

See Chapter 8 on machine performance for more information.

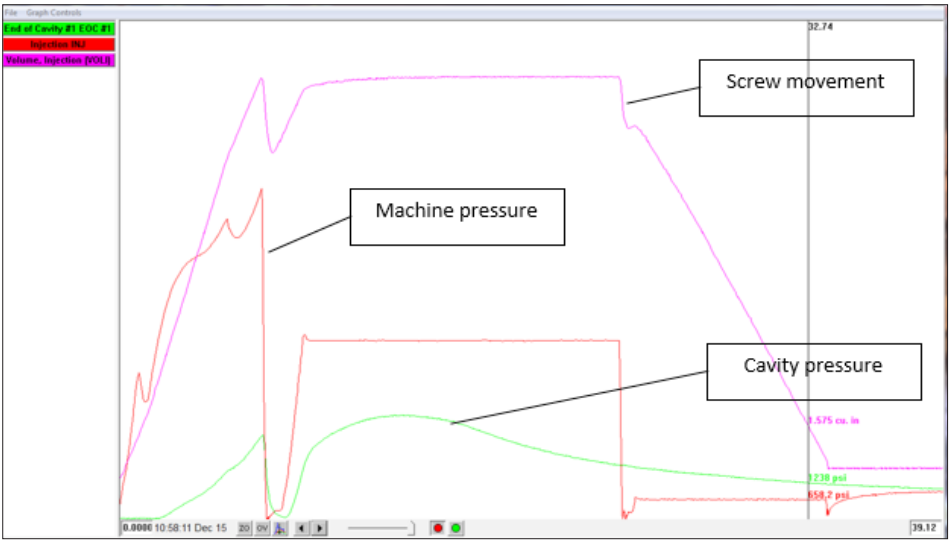


Figure 26.5 RJG eDART® graph showing poor velocity to pressure response

26.3.3.3 Machine: Non-Return Valve

To consistently inject plastic into the mold the non-return valve must react in a consistent fashion. If the non-return valve is not consistent there will be leakage through the valve during injection resulting in inconsistent shots and consequently inconsistent dimensions. If a non-return valve is leaking the cushion of the machine will either be inconsistent or bottom right out. One way to check for potential leakage of non-return valves is to watch the injection ram while injecting plastic because a leaking non-return valve will often leak enough plastic back through the screw flights that the screw will show rotation during fill. Also watch how much screw travel is occurring during hold; if there is a great deal of forward travel it may indicate issues with the non-return valve.

Many things can happen to a non-return valve including wear over time, breakage, and trapping foreign material. A non-return valve relies on steel-to-steel seating that will wear over time, especially if running glass-filled materials. Cracked non-return valves can be the result of a press being started up without adequate

preheat time, and there are even cases where the whole screw tip assembly has broken off the screw and is free floating in the barrel. Metal or unmelted plastic can become trapped in a non-return valve, which will prohibit the valve seating properly resulting in leakage. Be aware that damage or wear may not be obvious. Figure 26.6 shows a ball-style non-return valve that had to be replaced due to leakage.

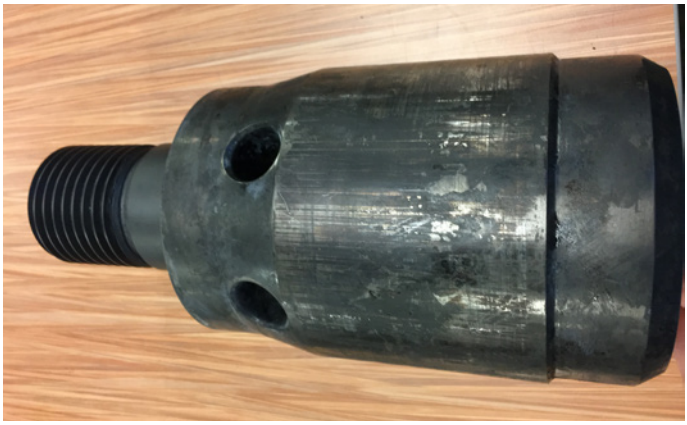


Figure 26.6 Ball check non-return valve

When investigating a potentially leaking non-return valve the best method to analyze is to run a study where 10 consecutive fill only shots are compared for consistency.

See Chapter 8 on machine performance for specific information on testing non-return valves.

Another potential factor could be that the barrel is actually worn and allowing leakage as opposed to an issue with the non-return valve. To verify that the issue is not in the barrel move the shot size and transfer position back one inch and see if the problem goes away. Because processes often have transfer positions set at the same point this location in the barrel may experience wear over time.

Keep in mind that adequate decompression must be used in the process to allow a sliding ring to function correctly. Before diving into tearing apart the machine verify that decompression is not the culprit.

26.3.3.4 Machine: Machine Performance

If a press is not capable of repeatably controlling the process to the desired set points the parts will have dimensional variation. A machine must accurately control all of the process variables to ensure that every shot matches what has been established as the process.

Figure 26.7 shows 10 consecutive shots with significant second-stage pressure and time variation. The part being molded in this machine was showing dimensional variation and needed frequent adjustments. A portable process monitoring system was put on the machine and it was quickly apparent that there was a major machine problem. In this particular case a valve on the molding machine needed to be rebuilt; after rebuilding the valve the problem was resolved.

See Chapter 8 on machine performance for more information.

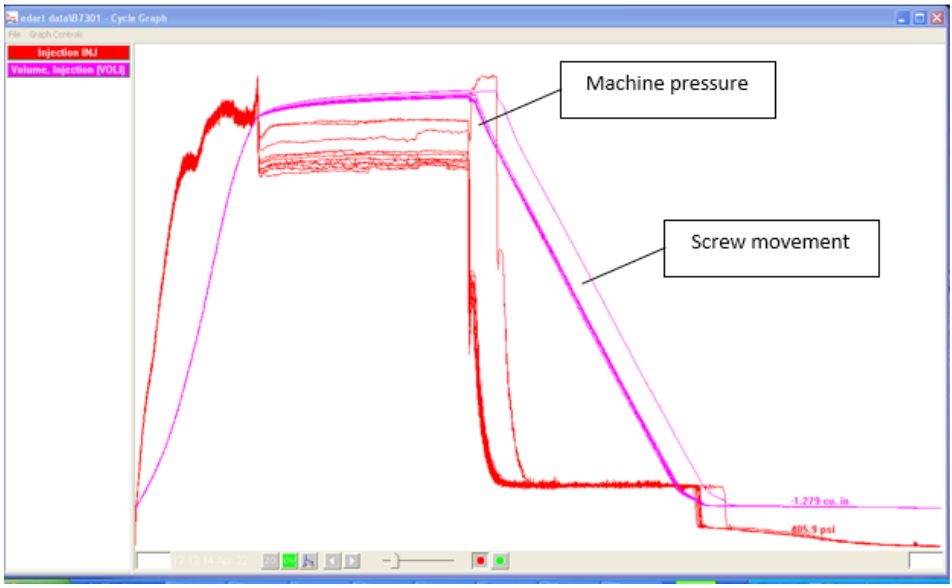


Figure 26.7 RJG eDART® showing 10 consecutive shots with second-stage pressure variation

26.3.4 Dimensions Troubleshooting Material Issues

As with any defect it is important to verify that the correct material is being used. In general, unfilled semi-crystalline materials will shrink more than filled and amorphous materials. If the part is consistently having a dimensional problem there are some things that may be worth evaluating for improvement. Items to consider if having dimensional issues are:

- Filled material
- Nucleating agents
- Material type

26.3.4.1 Material: Filled Material

A filled material will be more dimensionally stable and experience lower overall shrink than will an unfilled material. Experimentation with a filled version of the material may yield better results and produce a dimensionally acceptable part. This potential will require a great deal of effort between the molder and the customer to verify overall impact and suitability to the application but it may be the only choice other than building a new mold.

26.3.4.2 Material: Nucleating Agents

Addition of a nucleating agent to a semi-crystalline material will cause the material to crystallize faster, which can reduce some of the shrinkage of a semi-crystalline material. Use of a nucleating agent may impact the overall physical properties of the material so care should be taken to evaluate the overall impact of running the process with a nucleated material.

Various additives to the material such as glass and mineral will also create a nucleating effect on the material. If the filler content is varying the nucleating impact may also be varying and impacting the dimensions of the part.

26.3.4.3 Material: Type

It is important to understand that at a basic level the difference in shrinkage between an amorphous material and an unfilled semi-crystalline material can be very large. In some cases, an unfilled semi-crystalline material may shrink 4–5 times the amount of an unfilled amorphous material. See Table 26.2 for a basic list of material types.

Table 26.2 Basic List of Material Types

Amorphous	Semi-crystalline
Polycarbonate	PPO (Acetal)
PMMA (Acrylic)	PET
ABS	Polypropylene
Polystyrene	Nylon
ASA	PBT

The material type must be planned for ahead of time and the appropriate shrink factors for the chosen material must be used (STOP, Think). If a mold is cut to the wrong shrink factor it can be extremely difficult to get the part to the required dimensions. Sometimes a material change will occur during process development, which will result in trying to achieve dimensions even though the wrong shrink factor was used. This makes planning for shrink very difficult.

One of the challenges with plastics is that many factors can influence the shrink rate of the material, which is why suppliers always give a range of shrink rates. Some of the factors to consider when discussing shrink include:

- Wall stock
- Flow length
- Distance from gates for distributing pressure

The above factors relate back to part and mold design and may need to be compensated for. Another factor to keep in mind is that plastics will change size based on the temperature the part is exposed to. Even the difference between measurements in the hot factory environment as compared to an air-conditioned quality lab can produce results that are significantly different. If experiencing parts that do not match from summer to winter investigate the linear thermal expansion characteristics of the material. Some size limits will be set too tight to accommodate seasonal variation in size. The variation of plastic should be considered up front when designing parts as well as when dimensional tolerances are set.

Be aware of anisotropic shrink, which is differential shrink that occurs with flow and across flow. A challenging part of designing appropriate shrink numbers on plastic parts is knowing that anisotropic shrink may occur leading to cutting a mold with different shrink factors depending on the properties of the part. This concern can magnify when molding filled materials. Warp analysis on CAE can help provide insight into this area.

27 Excessive Cycle Time

27.1 Description

Parts are running with cycle times that are causing capacity issues, increased cost, etc.

27.2 Excessive Cycle Time Troubleshooting Chart

Table 27.1 shows the troubleshooting chart for excessive cycle time.

Table 27.1 Troubleshooting Chart for Excessive Cycle Time

Molding Process	Mold	Machine	Material
high mold temperature	lack of cooling	slow clamp speed	regrind
high melt temperature	damaged or plugged cooling lines	slow reaction times	additive package
timers set higher than required	gate size	slow automation	
slow fill speeds	runner and sprue size	ejection speed and strokes	
long cooling time	poor ejection		
	thick sections		
	thermal isolation of hot runner		

27.3 Excessive Cycle Time Troubleshooting

Many factors can impact the cycle time of a molded part. It is important to optimize the cycle of a mold during process development and make every effort to maintain that cycle for the life of the product.

27.3.1 Excessive Cycle Time Troubleshooting Molding Process Issues

Factors in the process that may lead to long cycle times include:

- High mold temperature
- High melt temperature
- Timers set longer than required
- Slow fill speeds
- Long cooling time

27.3.1.1 Molding Process: High Melt Temperature

The dynamics of injection molding thermoplastics means that energy is added to the plastic to melt the plastic and then after injecting the plastic into the mold we must pull that energy back out to allow the melted plastic to resolidify. The temperature of the molded part must drop to an appropriate temperature to allow the part to be ejected from the mold in a state that will retain its molded shape; if ejected too hot the part will warp and change shape as it continues to cool outside of the mold. Every BTU of energy that is put into the plastic during the melting stage must be removed during the cooling stage. It is important to remember that the parts can be ejected at a temperature that is often well above the mold temperature, and cooling will continue outside of the mold after ejection.

If the melt temperature of the plastic is on the high side of what is required, chances are that the cycle time will be increased due to longer cooling time and potentially longer time required for gate seal. A high melt temperature is a double-edged sword that costs money for the extra energy input into the machine to create it, and requires extra process time to cool back down from the high temperature.

The two typical things to consider when detecting a high melt temperature are:

1. Melt temperature is high for no good reason.

   This is often the case because a process was developed with a given set of barrel temperatures, screw recovery speed, screw style, and back pressure. All of these process settings will interact to create a melt temperature. Many molders will either completely skip checking melt temperature or check the melt temperature and upon seeing that it is high do nothing. As a result of not caring about

melt temperature the rest of the process will need to be dialed in to accommodate the melt temperature.

2. Melt temperature is run high to compensate for something else.

   A typical example of this is a situation where a part is designed without adequate gating, which in turn leads to a pressure limited machine as the part is filled. In such a situation a typical process adjustment is to raise the melt temperature to try to process a lower pressure to fill the mold.

To help avoid excess cycle time due to high melt temperature it is important to make sure that first of all people understand the impact of high melt temperature on cycle time. Also during part and mold design look for conditions like long flow lengths, thin wall sections, and lack of gating. If these part design issues are not addressed before steel is cut the process will be left to suffer.

Case Study: Nonuniform Wall Stock

In this part the nominal wall stock was thinner than certain areas with a much thicker cross section. As a result of the variation in nominal wall stock the material temperature was run on the high side of the material supplier's recommended temperature range. As a result, the cycle time to cool the thick areas was longer than necessary. A secondary problem with this type of part design is that the thinner nominal wall stock freezes early preventing full pack of the thick sections, which leads to sink, nonuniform cooling, and shrinkage resulting in warp.

Maintaining nominal wall stock is critical for quality molding processes!

If the melt temperature can be dropped without impacting part quality there is a good chance of reducing the cycle time for the part. If reducing the melt temperature causes other problems dig deeper and try to resolve them, which in turn will allow the part to be molded with a cooler melt temperature and potentially a faster cycle time.

27.3.1.2 Molding Process: High Mold Temperature

The mold acts as a heat exchanger to remove the heat from the melted plastic. Lack of cooling in a mold will lead to longer cycle time to allow the part to adequately cool. It is important to understand that one hot spot in a mold can drive the overall required cooling time for a part. Therefore, uniform cooling will lead to a better opportunity to minimize the cycle time for any molded part.

Verify that the thermolator is running to the specified set point. Surface temperature checks or thermal imaging of the temperature of the part exiting the mold may indicate cooling problems. Also surface temperature checks of the mold surface can indicate cooling-related concerns.

See Chapter 14 on mold cooling for advice on optimizing the cooling potential of a mold.

27.3.1.3 Molding Process: Slow Fill Speeds

The molding process is made up of a variety of times, one of which is fill time. Fill time is determined by the fill velocity used on the molding machine. In a normal Decoupled® II molding process the first stage of injection is referred to as fill, in which the mold will be filled to 95–98% full at as fast a speed as possible.

Some molds will require slow fill speeds for optimal product quality; for example, lenses are a case that often requires a slow fill speed. If there is no product-related reason that requires a slow fill speed then the part should be filled fast to both improve cycle time and help maintain consistent melt viscosity.

Do not process around other issues with fill speed, for example:

1. Burns

   Burns should not be processed around with fill speed because burning is an indicator of a lack of venting (see Chapter 7 on venting). Although slowing fill velocity will in fact usually impact burning, the process window is narrowed due to a simple tooling fix. Vent the mold and maintain fast fill speeds!

2. Flash

   Flash is not driven by velocity because flash is a result of high enough cavity pressures to overcome the force holding a mold closed. Filling fast does not flash the mold, but the sudden spike in cavity pressure that occurs if the mold is overfilled during fill will cause flash. If a machine is transferring well from velocity control to pressure control and the fill only shot is adequate the mold should not flash from fast velocities.

27.3.1.4 Molding Process: Long Cooling Time

Cooling time should be optimized to get the part out of the mold as fast as possible. Sometimes people will get caught in the “even number” syndrome where 20 seconds of cooling time seems like a better number than 19 seconds. Parts should be ejected as soon as possible without causing other issues such as warp.

Part out temperature after ejection should be documented during process development. Using a thermal imaging camera will provide an accurate picture of any hot spots and potential cooling problems.

If cooling time is running long due to long screw rotate times consider adjusting barrel temperature settings to determine if the recovery rate can be improved. Also review if the shot size is over 75% of the rated barrel shot capacity. Large shot sizes may limit the cycle time due to long recovery times. The mold may run faster in a press with a larger barrel capacity. Also watch for screw and barrel wear that can

lead to reduced recovery rates. If rotate time is increasing over time it can indicate a worn screw or barrel especially when running glass-filled materials.

For long cooling times driven by poor cooling of the mold see Section 27.3.2. Do not use the molding machine and mold as a cooling fixture! Look for opportunities to improve cooling to help minimize cycle time.

27.3.2 Excessive Cycle Time Troubleshooting Mold Issues

Mold-related problems include:

- Lack of cooling
- Damaged or plugged cooling lines
- Gate size
- Runner and sprue size
- Poor ejection
- Thick sections
- Thermal isolation of hot runner

27.3.2.1 Mold: Lack of Cooling

If a mold cannot remove the added heat from injecting molten plastic the cycle time will become a problem. All molds must be designed with optimized cooling and details such as cores should be cooled whenever possible.

See Chapter 14 for more information on mold cooling.

27.3.2.2 Mold: Damaged or Plugged Cooling Lines

If a cooling line in a mold becomes plugged water will be restricted and may not achieve turbulent flow through the circuit. Any lines that have become plugged or have restricted flow will lead to cooling problems. Also beware of blades that have been inserted parallel to water flow rather than perpendicular to flow. Also watch for cases where bubblers or blades have been inserted too far and are restricting water flow through the mold.

Verify flow through all circuits. A thermal image of the part can be useful for identifying areas with poor cooling. Verify that the mold is watered correctly; for example, Figure 27.1 shows a mold that has bubblers watered with the water flowing in the wrong direction through part of the bubblers leading to excessively high core temperatures.

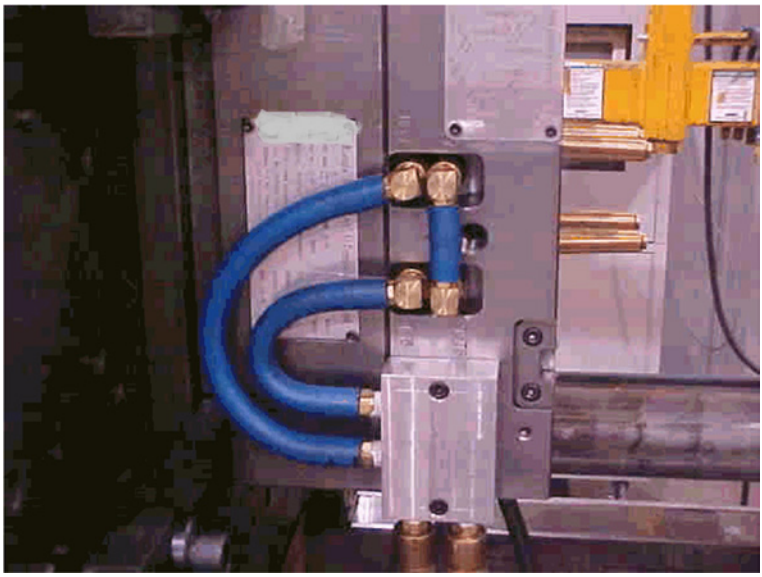


Figure 27.1 Pair of bubblers watered incorrectly: outlet of first bubbler goes to outlet of second bubbler rather than the inlet

27.3.2.3 Mold: Gate Size

When a mold has a gate that is excessively large the time required for gate seal will increase. There are many times when a gate size is designed to be too large for what is required to pack out the part. Many people rely on a variety of rules of thumb but recent research has shown that there are many cases where the old recommended gate size could be downsized leading to significant reduction in cycle time.

If the gate size is not influencing the molded part for size or sink reduction there is a possibility that it is oversized. There are many cases where starting with a smaller gate size may yield perfectly acceptable parts but also allows for a steel safe condition where the gate can be increased if it is too small.

27.3.2.4 Mold: Runner and Sprue Size

There are many examples in industry of excessive sprues or runners. They must be designed to provide adequate flow and pressure to the cavity but often people use the thought process that a bigger runner will always be better for the part. There are plenty of cases where a smaller runner does not increase fill pressures and still allows for adequate pressurization of the cavity. Thicker runners often need longer cooling times and often control the cycle time due to the extra required cooling time for the runners.

27.3.2.5 Mold: Poor Ejection

Ejection issues can lead to longer cycle times. The two most common problems encountered with ejection are:

1. Lack of ejection surface area leading to pin push.

   To process around this issue the cooling time is often increased. The mold should be designed with adequate ejection to ensure that the part will be pushed off efficiently during ejection. Bear in mind that materials with higher shrink rates will often require more ejection area for effective part removal. Also be sure to examine the part design for areas with poor draft, deep draws for ejection, and shrink on details, because all will demand more ejection area to push the part from the mold.

2. Extra ejection strokes.

   Sometimes a part will not eject well due to a variety of issues and extra ejection strokes may be added to ensure proper ejection. Each extra time the ejector strokes is increasing the cycle time. If the part is sticking the mold should be examined for improvements. In some cases the solution can be as simple as allowing increased injection stroke. Also keep in mind that every extra ejector stroke is increasing the wear on the mold and machine. Look for the reason for the extra strokes and eliminate the issue.

27.3.2.6 Mold: Thick Sections

Molds with thick wall sections will require additional cooling time. For most cases this must be addressed in part design (STOP: Think). Maintaining a consistent nominal wall stock is a fundamental design rule that cannot be overemphasized. A mold can be built with thick sections even where the part design does not call for increased thickness. A prime example of this problem is when a core pin is shorter than required, which leaves a thick section under the pin. If a part has sink opposite a core pin location check the depth of the core pin to ensure that it is long enough to avoid the risk of sink over the pin.

Also if a part wall thickness must be locally increased try to blend the wall stock rather than having an abrupt transition. This blend may allow less risk of sink and read-through (variation in gloss levels due to differential shrink) at the wall stock increase.

27.3.2.7 Mold: Thermal Isolation of Hot Runner

A hot runner system adds a large heat load to a mold that needs to be isolated from the cavity. If the heat in the hot runner is not well managed extra heat may transfer into the cavity leading to extended cycle times. The manifold should be thermally isolated and controlled to minimize the heat rise in the cavity. Watch for issues

where the hot tip contacts the cavity because there can be a large buildup of heat in this area that can lead to increased cycle times to control stringing from the heat.

27.3.3 Excessive Cycle Time Troubleshooting Machine Issues

Machine-related causes include:

- Slow clamp speeds
- Slow reaction times
- Slow automation
- Ejection speed and strokes

27.3.3.1 Machine: Slow Clamp Speeds

Over time injection molding machines will wear. Valve seals wear on hydraulic presses causing leakage in cylinders. The mechanical components on a machine can wear and become sloppy or loose over time. Both of these wear situations can lead to reduced clamp velocities.

If cycles are slowing over time and the time increase is coming from clamp opening and closing times it is time to calculate the amount of lost press time and money. Rebuilding the clamp of a machine can bring the clamp movement speeds back to a more normal rate.

Some machines have the ability to allow the clamp to decompress after completion of the second stage. This decompression of the clamp can allow a quicker response when the machine is ready to initiate clamp movement. Take a look at the machine controller to see if this option is available and utilize it where possible.

Also evaluate the amount of breakaway distance and clamp slowdown distances. Often there is time to be gained by optimizing where the clamp slows down and accelerates. The clamp should be running in a smooth fashion without banging, but the speeds should be optimized to limit the amount of time the machine spends moving the clamp. Long travel distances at slow speeds will add to the overall cycle time.

27.3.3.2 Machine: Slow Reaction Times

When looking for small gains in cycle time one area to examine is the amount of delay or hesitation at various stages of the process. Using a stopwatch it is possible to evaluate various time delays such as:

- Start of cycle to start of mold close
- Mold close to clamp lockup

- Clamp lockup to start of injection
- Cool time completion to start of mold open
- Mold open to start of ejection

All of these signals and movements can be instrumented and collected with a data acquisition unit. By taking cycle time analysis to this level it is possible to start shaving non-value-added time from the process.

27.3.3.3 Machine: Slow Automation

Watch for slow robot movements because clamp open time costs money. If robot velocities are not being optimized this may be an opportunity for cycle time reduction. Look for opportunities to use a "pounce" position where the robot is prepositioned closer to the mold. Another useful technique is using sweeping motions rather than XYZ point-to-point movements.

Also watch for delays in the amount of time for the robot to build vacuum. Shorter vacuum lines and larger diameter lines may allow vacuum to build quicker. Look at potentially adding a larger vacuum pump or additional vacuum pumps. There are even systems to locate a vacuum pump at every vacuum cup to minimize the distance the vacuum has to draw.

Does the product even need a robot? If the part can be ejected onto a conveyor without damage the time required for robot movement is eliminated. Do not assume because the machine has a robot that it always has to run for every mold.

27.3.3.4 Machine: Ejection Speed and Strokes

If a mold is being run with multiple ejector strokes ask why? Each extra ejector stroke costs cycle time. Also, if the machine stroke is longer than required for the mold, use adequate stroke for part removal but not extra.

Just like with the clamp of a machine the ejector cylinders can wear and leak resulting in reduced ejection velocities.

Using short knock-out bars and returning the ejector plate all the way will create wasted time as the ejector strokes the distance that is not engaging the mold.

27.3.4 Excessive Cycle Time Troubleshooting Material Issues

In general, some materials will provide faster cycle times than others. If running into variation in effective cycle time due to material issues some of the possible causes may be:

- Regrind
- Additive package

27.3.4.1 Material: Regrind

Sometimes when running regrind the pickup rate of the screw will vary. This occurs sometimes because of regrind bridging the feed throat or not feeding well through the feed lines. When the material is lacking the screw will be starved of material and will continue to run until a full shot is built. Another impact of this extra screw run time can be a change in melt temperature of the plastic in the barrel during the increased rotation time of the screw. Watch how material is feeding the machine and verify that the clamp opening is not waiting for the screw to recover.

27.3.4.2 Material: Additive Package

There are a variety of factors in the formulation that can impact the cycle time capability of a material. One example is a variation in nucleating agents in semi-crystalline materials. The higher the level of nucleating agents the faster the cycle time can be. Another additive impact could come from variation in lubricants and/or release agents because these compounds can influence how the screw is feeding material during plastication.

If the material is the root cause of needing to increase cycle time there will normally be an effect when changing into a new lot of material. To determine if the material is contributing to the problem try swapping to another lot of material or even try sampling another similar material to determine impact.

28 High Fill Pressure

28.1 Description

High fill pressure can lead to other additional problems including a pressure limited molding process. If a molding process is pressure limited in a molding machine the machine will not be capable of maintaining consistent fill velocities. If a process has been running and the fill pressure suddenly increases investigation into the root cause should be undertaken.

28.2 High Fill Pressure Troubleshooting Chart

Table 28.1 shows the high fill pressure troubleshooting chart.

Table 28.1 High Fill Pressure Troubleshooting Chart

Molding Process	Mold	Machine	Material
low melt temperature	gate size and number of gates	nozzle tip style or orifice	viscosity increase
fill velocity	runner or sprue size	nozzle style	moisture content
heavy fill only	hot runner issues	plugged tip or nozzle	
	plugged gates	machine performance	
	venting		
	wall stock		

28.3 High Fill Pressure Troubleshooting

If a molding process suddenly is experiencing higher than normal fill pressures it is critical to determine what has changed. If the fill pressure is much higher than normal it will indicate that the pressure drop across the cavity is also greater than normal, which means the plastic is not going to be packed like it was previously.

If the fill pressure reaches the maximum available for the molding machine the process is considered to be pressure limited. When a molding process is pressure limited the machine cannot control the fill velocity. Molding machines must have more pressure available than is required to maintain control of fill velocity. Bear in mind that processes that are within 10% of the maximum pressure of the machine are on the edge and will likely have molding runs that are actually pressure limited due to changes in material viscosity. When establishing a molding process make sure that the required fill pressure is less than 90% of the maximum pressure of the molding machine.

28.3.1 High Fill Pressure Troubleshooting Molding Process Issues

Possible molding process causes are:

- Low melt temperature
- Fill velocity
- Heavy fill only

28.3.1.1 Molding Process: Low Melt Temperature

Fill pressure typically has an inverse relationship to melt temperature, especially when dealing with amorphous materials. The lower the melt temperature the higher the pressure required to fill a mold. There can even be conditions where the melt temperature is so low that the mold will not fill completely before the melt front freezes off.

If the process is experiencing a sudden increase in peak pressure a melt temperature check may provide the root cause. Verify that the actual melt temperature corresponds to the documented melt temperature on the setup sheet. If there is no documented baseline melt temperature for the process at least verify that the melt temperature agrees with the supplier's recommended values.

If the melt temperature is determined to be low the following items should be investigated:

- Set points for barrel temperatures
- Set points for back pressure and screw rotate speeds

If the set points match the documented process then see Section 28.3.3 on machine troubleshooting for further information on possible causes for low melt temperatures.

28.3.1.2 Molding Process: Fill Velocity

Verify that the fill time matches the documented process. If the mold is being filled at a fill velocity that does not match the documented process the fill pressure will be different. Keep in mind that as long as the process is not pressure limited the fill velocity will be maintained by using whatever pressure is required. Fill time should be repeatable shot after shot without fail because variation in fill time will create other potential part defects.

28.3.1.3 Molding Process: Heavy Fill Only

Transferring from first stage to second stage should occur at a 95–98% full cavity. If the machine is transferred late (transfer position set too low) and the cavity is completely filled, the sudden resistance to flow will cause a very rapid spike in fill pressure.

Fill only weight should always be checked when starting up a process and troubleshooting. An incorrect fill only weight will lead to a wide variety of potential defects. A process that transfers late will be much more likely to flash the mold. Remember that fast fill velocities do not flash a mold but the sudden spike in cavity pressure does if transferring too late.

28.3.2 High Fill Pressure Troubleshooting Mold Issues

Potential mold-related causes include:

- Gate size and number of gates
- Runner or sprue size
- Hot runner
- Plugged gates
- Venting
- Wall stock

28.3.2.1 Mold: Gate Size and Number of Gates

If a gate is too restrictive the pressure required to force the plastic through the gate may increase. This is not as hard and fast of a rule as many people would prefer it to be. There are cases when a smaller gate size has minimal impact on the actual fill pressure required for the mold. It is important to understand that smaller gates

produce a higher shear rate of material during injection, which in turn may lead to pressures staying low.

The total area of gates is important to consider. A wider gate may help with more flow volume without increasing the thickness and impacting gate freeze time. Also review if there are enough gates to adequately fill the part. Every material has a maximum flow length to wall stock ratio that needs to be considered. If trying to flow the material across a long part the pressure will climb throughout fill, but adding a second gate could cut flow length in half. Examine the gate location for ideal fill: a gate located at the mid-point of the part cuts flow length in half when compared to a gate located at the end of the part.

Long gate lands are an often-overlooked area. Long gate lands will restrict the flow of the plastic and lead to increases in fill pressure. Land length should never exceed 0.030 in on a gate, and dropping it to 0.005 in is often a very effective solution for a variety of problems including high pressure.

Keep in mind that gate size will also influence the gate seal time and the ability to pressurize a mold effectively.

28.3.2.2 Mold: Runner or Sprue Size

It is important to understand that the gate is not the only restriction in the material feed system. The runner and sprue can have undersized thickness for a given part, which in turn can lead to an increase of the required pressure to fill the mold.

Long runners and sprues will also impact the fill pressure of a mold. The further the plastic flows through a sprue and runner prior to hitting the cavity the higher the pressure will be. Whenever possible minimize the length of the sprue and runner to minimize the pressure required to fill the mold. In general, longer runners will require a thicker runner cross section.

28.3.2.3 Mold: Hot Runner

If there are restrictions within the hot runner system the pressure required to fill the mold will rise. These restrictions can appear through the hot runner system. The most common restriction is at a low-vestige-style hot tip. Low-vestige hot tips can be a much smaller restriction then the downstream runner and gate and lead to excessive pressures. Hot tips should be evaluated for restriction to the flow and sized accordingly.

Another factor with hot runner systems can be zones that are not heating properly leading to cold zones that cause premature freezing of the material. The hot runner zones should be set so that they neither add nor remove heat from the melted plastic. Hot runner zones should normally be set at the actual melt temperature of the material.

Also keep in mind that restrictive hot runner tips can become plugged with contamination such as alternate materials, metal, or other foreign materials. There have been many hot runners that needed to be torn apart to have contamination cleaned out. In some cases even the internal spreader tip on the hot runner can break loose and plug the orifice.

28.3.2.4 Mold: Plugged Gates

Whether with hot drops or standard gates there can be spikes in pressure due to material plugging the gate. Sometimes partially melted pellets will make their way to the gate and create a spike in pressure before they bust through the gate into the cavity.

If a process is experiencing intermittent spikes in pressure check for melt consistency. Sometimes when using a large percentage of the barrel the melt will be of poor quality, especially when running a semi-crystalline material. Unmelted pellets plugging gates will many times lead to short shots but they also may lead to high fill pressures.

Another possibility for plugging gates or drop tips is when a piece of frozen material does not get trapped in a cold slug well. These frozen cold slugs can easily plug a gate or drop orifice until the pressure reaches a point that the cold slug is forced through the tip.

28.3.2.5 Mold: Wall Stock

Thin wall stock can lead to high fill pressures. In recent years there has been a push to have more injection pressure available on molding machines to fill thinner wall stocks. Check with the material supplier to confirm that the wall stock to flow length is acceptable. Early CAE studies will help determine if a part can be filled.

28.3.3 High Fill Pressure Troubleshooting Machine Issues

Machine-related concerns are:

- Nozzle tip style or orifice
- Nozzle style
- Plugged tip or nozzle
- Machine performance

28.3.3.1 Machine: Nozzle Tip Style or Orifice

The same tip style and orifice must be used for every run with a given mold. If a nylon-style tip is used instead of a full taper tip there will be a very different pressure drop through the tip. The extra restriction in a nylon-style tip will drive up fill

pressures. Also if a mold requires a 3/8-in nozzle orifice to match the sprue opening that is the tip that must be used, or otherwise again extra pressure drop is added to the system, which will impact how the material fills and packs.

Some molding shops will require a tip change for every mold change to ensure that this step is not neglected. One trick is to tack-weld a 7/8-in nut onto the mold to act as a tip retainer; this will provide a visual check to see if the tip has been removed from the side of the mold and hopefully installed on the machine.

Investigate methods for identifying tips when installed on the molding machine. This type of approach will make it easier to determine if a nylon tip or a full taper tip is on the press after the stamped markings have been "carboned up". Also if the tip is marked 360 degrees around it will always be easier to see directly when the purge guard is opened.

Another factor to keep in mind is that nylon tips are much more restrictive than a full taper nozzle with the same orifice. A nylon-style tip has a long taper from the orifice to the break point, which leads to a much smaller minimum diameter that the plastic must flow through. If a nylon tip is installed instead of the required full taper or general-purpose tip the increased restriction will lead to higher fill pressure as well as higher shear rates through the restriction. Figure 28.1 shows how the nozzle orifice tapers to a restriction depending on the tip style.

Figure 28.1 Cross section of general-purpose, nylon, and full taper nozzle tips (left to right)

28.3.3.2 Machine: Nozzle Style

In plants that use mixing nozzles there is opportunity to leave a mixer on the press when changing to a job that does not require it. The increased pressure drop through a mixing nozzle can be very large; for example, see Figure 28.2 for actual data comparing a general-purpose nozzle, a mixing nozzle, and a mixing tip. The

transfer pressure jumped up when moving to a mixing nozzle and mixing tip and the end of fill cavity pressure dropped significantly.

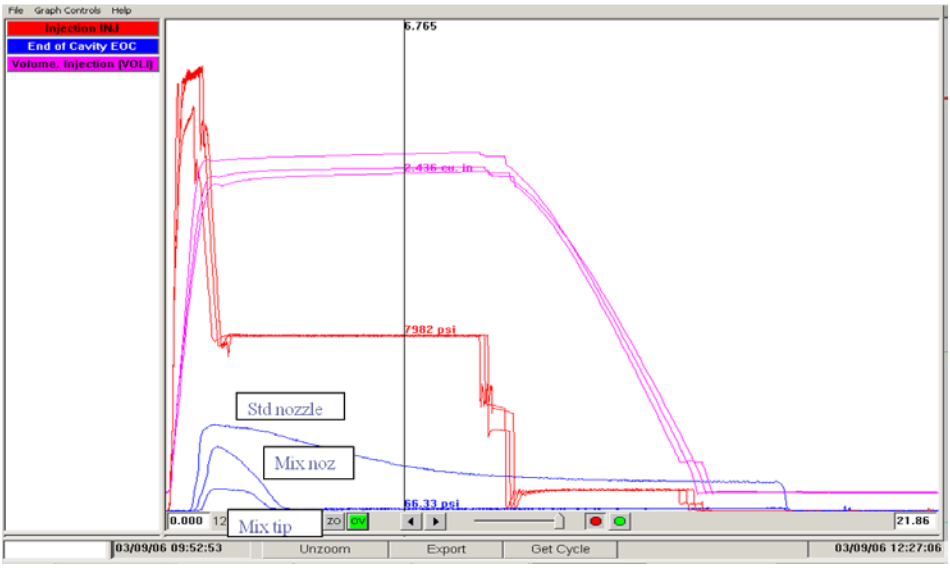


Figure 28.2 RJG eDART® graph showing three shots with different nozzles

A method for marking mixing nozzles should be implemented in any plants that run them. Running processes with a mixing nozzle not only raises the peak pressure to fill but also impacts shear on the material and packing pressure drop, and creates locations to trap contamination. They can be a useful tool to resolve color mixing issues but are really a "Band-Aid" to try to improve poor melt quality from an inferior screw.

Be aware of nozzle lengths as well. If a 6-in nozzle is all that is required, a 12-in nozzle should not be used as a "universal" nozzle. Longer nozzles create more restriction and potential pressure drop. It is also more difficult to maintain good thermal control on longer nozzles.

28.3.3.3 Machine: Plugged Tip or Nozzle

In everyday molding it is possible for nozzles or tips to become partially or totally plugged. If a nozzle or tip is partially plugged the peak fill pressure will increase due to the restriction. Excessive pressure drop through a nozzle can easily be detected by purging a shot from the machine. Things that can cause a plugged nozzle or tip include:

- Metal in a variety of fashions. In plants that mold metal inserts and use regrind, metal can contaminate the regrind when parts are ground with inserts. Also

when doing machine side assembly with fasteners there is a chance that a fastener could be dropped into an open material container. Magnets will not catch everything but they should be used to catch what they can.

- Unmelts. Unmelts are pellets that make their way through the screw and barrel without actually melting. In some cases running a semi-crystalline material like nylon or acetal will pass unmelts through the screw potentially plugging the nozzle or tip. Another case is where a material of higher melting temperature such as nylon contaminates a lower melting temperature material like polypropylene.
- Dispersion tip. If running with a dispersion tip there is already going to be a large increase in pressure. The other concern with a dispersion tip is that it will trap items in the holes further restricting flow.

Case Study: Intentional Sabotage

In this case the machine started running short shots, and the fill pressure became pressure limited. Troubleshooting through the problem showed that the purge pressure on the machine was very high. The nozzle tip was removed and a distorted penny was found jammed against the tip. The penny was blocking much of the material flow through the nozzle. The guess was that a disgruntled employee had decided to toss a penny in the material gaylord.

In another case a similar problem occurred and upon removing the nozzle assembly screws were found in the nozzle and tip.

28.3.3.4 Machine: Machine Performance

It is always critical to understand if a molding machine is actually achieving the process set points that are input into the controller. Just assuming that the machine will achieve a set point can be dangerous. The need for verification on what the machine is actually doing is why it is so important to document as many plastic condition outputs as is possible.

By documenting the fill time of a part, it really does not matter what the set value for fill speed is. A much better measure for fill rate would be volumetric flow rate because that would apply regardless of the screw size. For process consistency, repeating the documented fill only weight at the documented fill time means that the mold was filled with the same flow rate.

If the process is set at 500 psi hydraulic pressure, is that what the machine is achieving? Every press should be providing output that corresponds to the inputs.

See Chapter 8 on machine performance for more information on evaluating machine performance.

Case Study: Machine Accuracy

In this example a molding machine had a deviation of 200 psi hydraulic pressure from the set point. In this particular case the set point on the controller did not match the actual pressure displayed on the controller, nor did it match the hydraulic pressure gauge. An RJG eDART process monitoring system was connected to the machine to provide an additional confirmation and showed that the machine was in fact running 200 psi low on the hydraulic pressure. A handheld pressure gauge can be plugged into the test ports on the machine to verify pressure if a process monitoring system is unavailable.

This pressure inaccuracy explained why mold ABC never ran well in this machine! The process was not matching the documented and validated setup.

Another key point for machine control is how well the machine is controlling the temperatures on the barrel and nozzle. See Figure 28.3 for an example of a machine nozzle running with burned out heater bands. Review of thermocouple location is another item that should be considered, because often times end cap, adapter, and nozzle of a machine will be set up with a lack of thermocouples, which will impact thermal uniformity. Watch the duty cycle of heaters on the machine controller, because a zone that is always calling for heat or never calling for heat may indicate a zone that is burned out or has crossed thermocouples.

Remember that just because the process was set up correctly does not mean that the process conditions are being accurately applied. Machines will wear over time, hydraulics can leak, transducers give bad readings, wrong thermocouples may be used for replacement, and calibration can be off for any item, all of which can lead to lack of control.

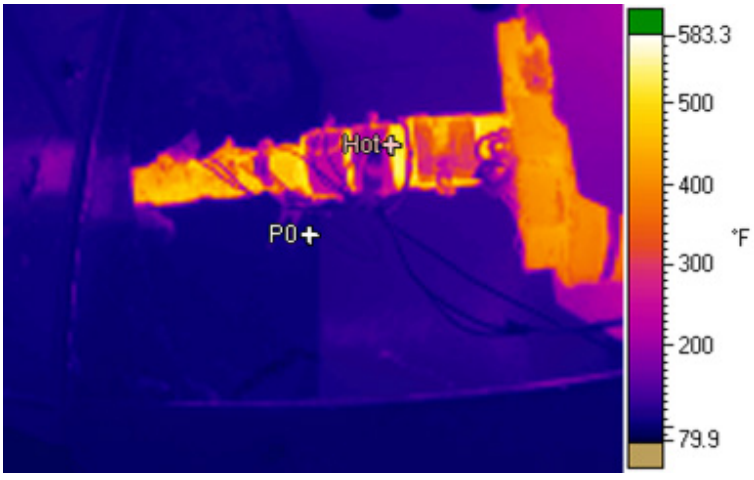


Figure 28.3 Thermal image of a long-extended nozzle with two heater bands not working

28.3.4 High Fill Pressure Troubleshooting Material Issues

Material causes can include:

- Viscosity increase
- Moisture content

28.3.4.1 Material: Viscosity Increase

Materials will have natural variation over time. As the viscosity of a material increases, the pressure to fill a mold will increase. This increase in viscosity may come from a change in average molecular weight of material, a change in filler content, or a change in additives. Every material will vary, but some materials will tend to have a wider range of variation than others. Melt index measurements are the typical measure used in industry to evaluate a material; however, it is important to understand that the melt index is determined at extremely low shear rates that do not simulate molding shear rates.

Changes in peak pressure on the press can be a good indicator that the viscosity of a material has changed. Higher viscosity means harder flowing, which means higher pressures. Viscosity is a measure of the resistance of a material to flow; higher viscosity means that the material flows with more resistance.

Case Study: Fill Pressure Increase with Barrel Size Change

This particular case was molding a PC/ABS part in a 170-ton press with a small barrel. The mold was moved into a 330-ton press to evaluate flash. Even though all the process documentation was followed for barrel temperatures, recovery time, etc., the fill pressure was substantially higher in the larger machine barrel. All intensification ratios were taken into account but the fill pressure was still higher in the larger machine. Further investigation showed that due to the shear rate during plastication the material was not achieving the same viscosity from barrel to barrel. The larger screw was turning slower and not generating as much shear heating during recovery. Remember that it is important to account for shear differences between different screw sizes.

28.3.4.2 Material: Moisture Content

In many materials moisture content during processing will cause hydrolysis of the material. Hydrolysis will cause a reduction in molecular weight of the material, leading to shorter molecule chains. With shorter molecule chains the viscosity of the material will drop and it will flow easier. This can create situations where there are large changes in peak fill pressure due the variation in viscosity that the moisture content is causing.

A common occurrence is to develop a process during the humid summer months with the material having some moisture when processing. When winter comes and the humidity drops the material will have a natural tendency to be dryer and as a result will require higher fill pressures than the wetter summer-processed material.

Nylon is very susceptible to moisture-induced viscosity variation. Because nylon can be run over a wide (relatively speaking) range of moisture contents the difference between 0.05% and 0.2% can translate into a large difference in fill pressure.

For more information on drying see Chapter 9.

29 Flaking

29.1 Description

Flaking is a defect where a small piece of plastic is shaved from a part and in turn leads to a surface defect on the part. Flaking often comes during automatic degating with gates such as sub gates and cashew gates, or from flash on pins, lifters, or slides. Figure 29.1 shows a magnified photo of a flake that was molded onto the surface of a part.

Also known as: cold slugs

Mistaken identities: flow line, splay

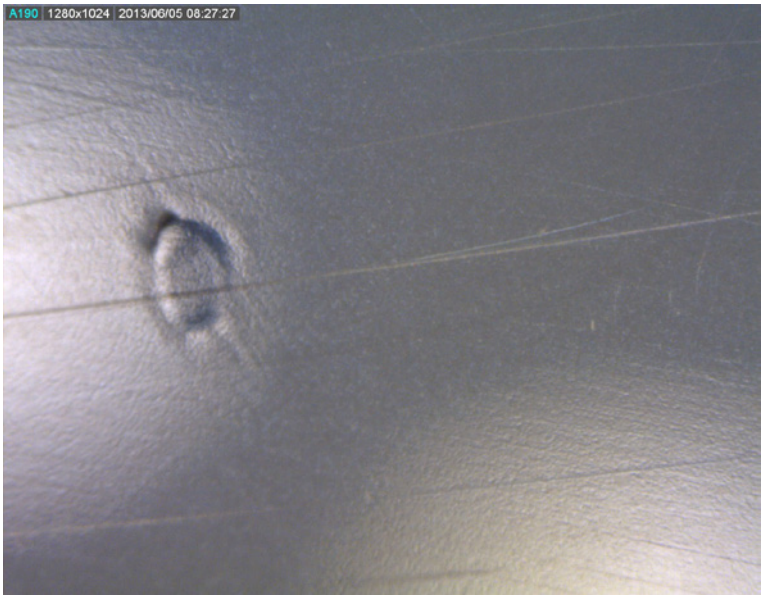


Figure 29.1 Close up photo of surface with a molded in flake

29.2 Flaking Troubleshooting Chart

Table 29.1 shows the flaking troubleshooting chart.

Table 29.1 Flaking Troubleshooting Chart

Molding Process	Mold	Machine	Material
damage during ejection	lifter, ejector pins, or slides	machine performance	
	gates		
	parting line burrs or damage		

29.3 Flaking Troubleshooting

The vast majority of flaking has a root cause in a mold issue. When troubleshooting what may be a flaking problem be sure to inspect the mold surface for flakes. If plastic flakes are evident on the mold surface the odds are good that they are also getting into the cavity and may be the root cause of the defect.

29.3.1 Flaking Troubleshooting Molding Process Issues

The molding process typically does not cause flaking concerns. Verify that there are no ejection-related problems occurring. Sometimes if the ejectors are running too fast they may cause some damage to the part that could lead to flaking. Also verify that the ejectors are not pulling back slightly before the part is extracted because this can pull the part back onto lifters leading to damage.

29.3.2 Flaking Troubleshooting Mold Issues

The mold is almost always the cause of flaking. The common sources of the flaking defect are:

- Lifter, pins, or slides
- Gates
- Parting line burrs or damage

29.3.2.1 Mold: Lifter, Pins, or Slides

One of the typical sources of flaking is flash that can occur around mold components such as lifters, ejector pins, or slides. When there is flash around any mold component the movement of the component during ejection can peel off the flash and leave behind a flake of plastic.

Another area of concern with mold components is when there is an undercut on a slide or lifter that can be peeled off during ejection. These areas of concern on a part will be highlighted by a scrape on the part. Where a part is scraped during ejection, plastic flaking can occur.

Any spots on the mold that are flashing must be corrected. If mismatch on a lifter or slide is causing the flaking the component must be spotted to eliminate the undercut.

29.3.2.2 Mold: Gates

Gates are another key area where plastic flaking occurs. When a sub gate or cashew gate breaks away from the parts, flaking from an unclean break can be a problem. The gate can break in a fashion that leaves behind a flake at the separation point; see Figure 29.2 for a magnified picture of a sub gate with flaking.

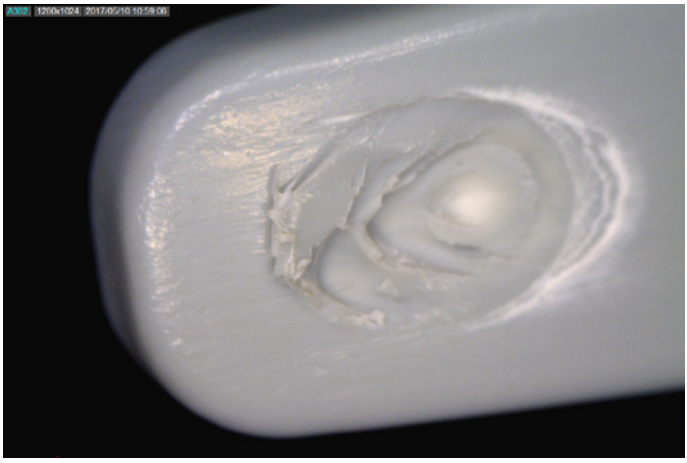


Figure 29.2 Magnified photo of sub gate with flaking

Often times a sub gate or cashew gate will have to have the break point sharpened to help provide a cleaner break. There will be situations where due to part quality constraints a sub gate or cashew gate will not be the best choice for gating due to risk of flaking.

A runner can have flash that can separate during ejection and leave behind flakes. Figure 29.3 shows a runner with excessive flash that led to flaking defects. Utilize

the STOP method to observe all of the potential flaking locations on the molded part, runner, and gate.

Figure 29.3 A runner with flash that will lead to flaking defects

Too much mass at a sub gate or cashew gate can lead to flaking where the plastic is not breaking clean. A "D" style sub gate will help eliminate this problem. See Chapter 4 on gating for more information.

29.3.2.3 Mold: Parting Line Burrs or Damage

Damage to the parting line of the mold can cause flakes of plastic to break away as the mold is removed from the cavity. The burr or parting line damage acts as an undercut that the plastic must move past and the result is often a flake separating from the part.

29.3.3 Flaking Troubleshooting Machine Issues

See Chapter 8 on machine performance.

29.3.4 Flaking Troubleshooting Material Issues

Specific materials will have more of a tendency to flake then others. Alloys will often have separation at the gate that can lead to flaking. Soft materials will often deform rather than shave past a burr or mismatch. Normally issues with the material are not the root cause of flaking.

30 Flash

30.1 Description

Flash is excess plastic that has extended out past the edge of the part. See for an example of flash.

Also known as:

Mistaken identity: mismatch

Figure 30.1 Flash

30.2 Flash Troubleshooting Chart

Table 30.1 shows the flash troubleshooting chart.

Table 30.1 Flash Troubleshooting Chart

Molding Process	Mold	Machine	Material
high second-stage pressure	parting line damage	tonnage	viscosity decrease
heavy fill only weight	vent depth	clamp parallelism	moisture content
clamp tonnage	trapped plastic	core pressure	regrind
melt temperature	support	mold size	
velocity to pressure transfer	erosion	toggle wear	
	slide deflection		
	mismatch		
	cavity balance		

30.3 Flash Troubleshooting

Flash is avoidable if three conditions are met:

1. The clamp pressure must be greater than the cavity pressure.
2. The mold must be robust enough to avoid deflection both in-line with clamp force and perpendicular to clamp force. See for a picture of force directions.
3. All shut offs and parting lines must be true net, meaning they are tight with no gaps, damage, or anything holding the mold open.

If flash is a concern use the STOP method to work through which of the above conditions are not being met.

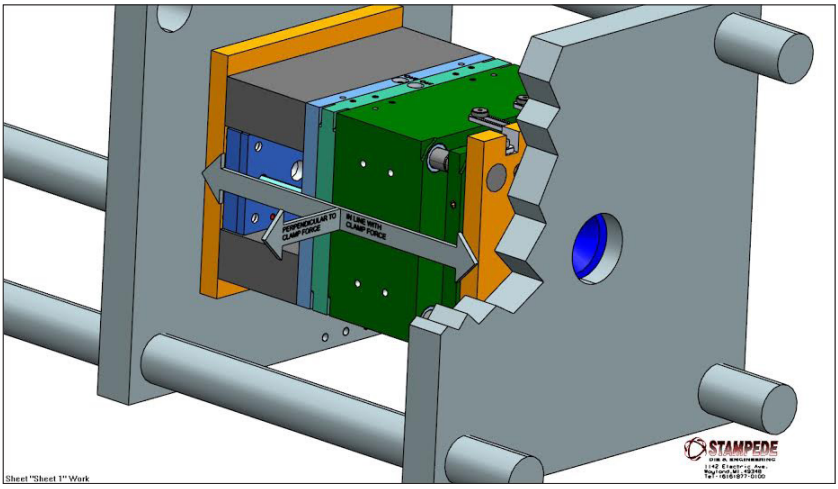


Figure 30.2 Forces act both parallel and perpendicular to clamp force

30.3.1 Flash Troubleshooting Molding Process Issues

Possible molding process concerns are:

- High second-stage pressure
- Heavy fill only weight
- Clamp tonnage
- Melt temperature
- Velocity to pressure (V to P) transfer

30.3.1.1 Molding Process: High Second-Stage Pressure

Second-stage set pressure is the pressure used to pack out the part and compensate for the shrinkage that occurs during cooling. If the pressure in the cavity overcomes the force being applied by the clamp unit flash will occur.

There will be cases where high second-stage pressure is used to either pack out a sink or try to make a part dimensionally larger. If a mold was cut with the wrong shrink factor, raising pressure can sometimes help to make the part larger; however, if the pressure becomes too high a larger tonnage machine may be required. Cavity pressure will act across the projected area of the mold but will not be uniform across the whole mold. Using average cavity pressure can give a prediction of the amount of force that is acting on the projected area; if this force becomes higher than the clamp force the clamp will be forced open slightly allowing the plastic to leak.

Verify that the set pressure matches the established process. Also verify that the actual pressure exerted by the machine is equal to what is specified in the documented process.

30.3.1.2 Molding Process: Heavy Fill Only Weight

Using RJG Decoupled Molding® techniques requires transferring from first-stage fill to second-stage pressure at a 95–98% full cavity. This is referred to as a fill only shot. Fill only weight should be documented and matched every time the mold is run. If the fill only weight is heavier than normal, i.e. filling further than 98%, there will be a good likelihood that the mold will flash. Filling a mold to 100% will cause a very rapid rise in cavity pressure that can overcome the clamp force resulting in flash. shows a simple plaque that was flashed from transferring too late.

Figure 30.3 Part flashed from transferring too late

To check the fill only shot, set second-stage pressure to zero or minimum allowed on the machine controller. Some controllers will also need to have second-stage volume, speed, or velocity turned off to completely stop any impact from second-stage pressurization. The resulting short shot should be weighed and compared to the documented fill only weight. If the part is too heavy the transfer position should be increased to put less plastic into the cavity during first-stage fill. Also examine the part at this point for flash because a fill only shot with flash is a good indicator of a tooling issue (see Section 30.3.2).

30.3.1.3 Molding Process: Clamp Tonnage

The clamp unit of the molding machine must hold the mold shut during the molding process. If the clamp force is too low then the cavity pressure will cause the mold to blow open allowing plastic to escape as flash.

Verify that the die height on a toggle machine is locking up tight. Many new presses have automatic die height settings that make this simple. On older machines the die height will need to be manually adjusted to ensure that the clamp is tight. On machines that have adjustable clamp tonnage settings verify that the tonnage is set to the appropriate level. A 500-ton press with the clamp tonnage set at 400 tons is no better than a 400-ton press. If you need the 500 tons make sure that the machine is set appropriately.

To determine if a mold is being forced open during the process use either a dial indicator or a 0–10-V device connected to a process monitoring system. The measurement device should be mounted across the parting line of the mold and zeroed out. Run the process on cycle and watch for movement.

The advantage to using a process monitoring input is that it allows a clear visual of what the machine injection pressure is doing when the mold actually starts to be forced open. In a process monitoring screen is shown with the “flat” curve representing mold deflection. It is clear to see that the mold starts to move open right as the machine transfers from first-stage velocity control to second-stage pressure control. This could indicate filling beyond 99%, which will result in a rapid spike in cavity pressure potentially resulting in mold deflection.

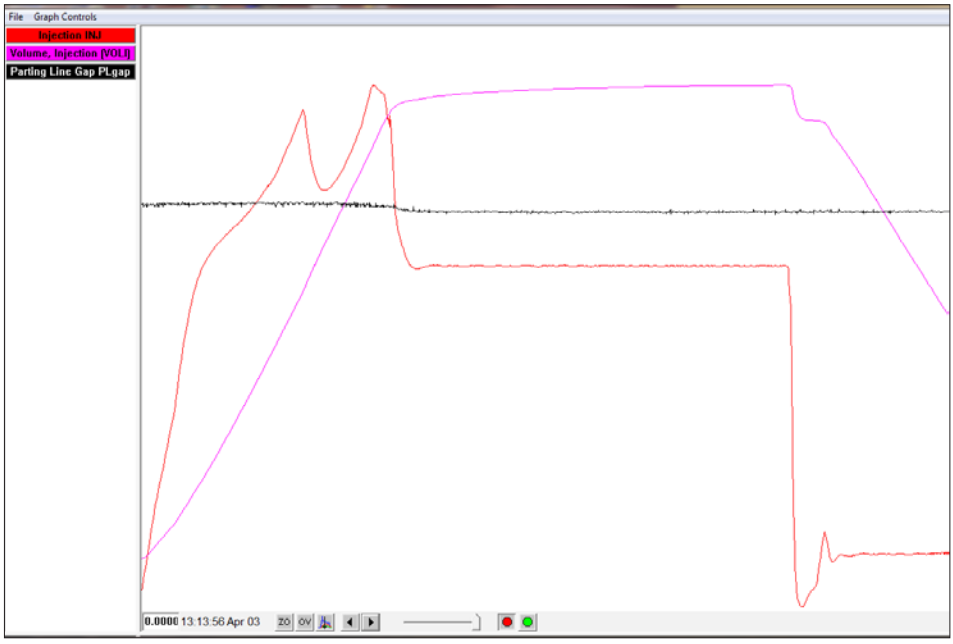


Figure 30.4 Process monitoring curves showing mold deflection

30.3.1.4 Molding Process: Melt Temperature

If the melt temperature of the plastic is running high the reduced viscosity of the plastic may allow the material to flash easier. Low-viscosity material will flow through thin cross sections easier even when the thin cross section is a gap in the parting line.

Low melt temperature will require higher pressures to fill the mold, possibly generating enough cavity pressure to force the clamp open.

Check melt temperature for comparison to the documented setup. If the melt temperature does not match the documented process then the following settings should be verified:

- Barrel temperature set points
- Back pressure
- Screw recovery speed

30.3.1.5 Molding Process: Velocity to Pressure Transfer

The way the machine transfers from first-stage velocity control to second-stage pressure control is an often overlooked cause of flash. If the machine spikes over the set pressure during transfer a resulting spike in cavity pressure will occur. This spike in pressure can lead to flash. depicts an RJG eDART® graph showing a poor pressure response. The sudden spike in pressure can easily flash a mold.

Case Study: Velocity to Pressure Flash

In this case study the part was being molded out of a mineral-filled TPO. The velocity to pressure transfer was causing a spike at the start of second-stage pressure, which over-pressurized the cavity resulting in parting line flash. The hold volume percent was adjusted and the spike at the velocity to pressure transfer point was almost completely eliminated. The resultant parts had no flash.

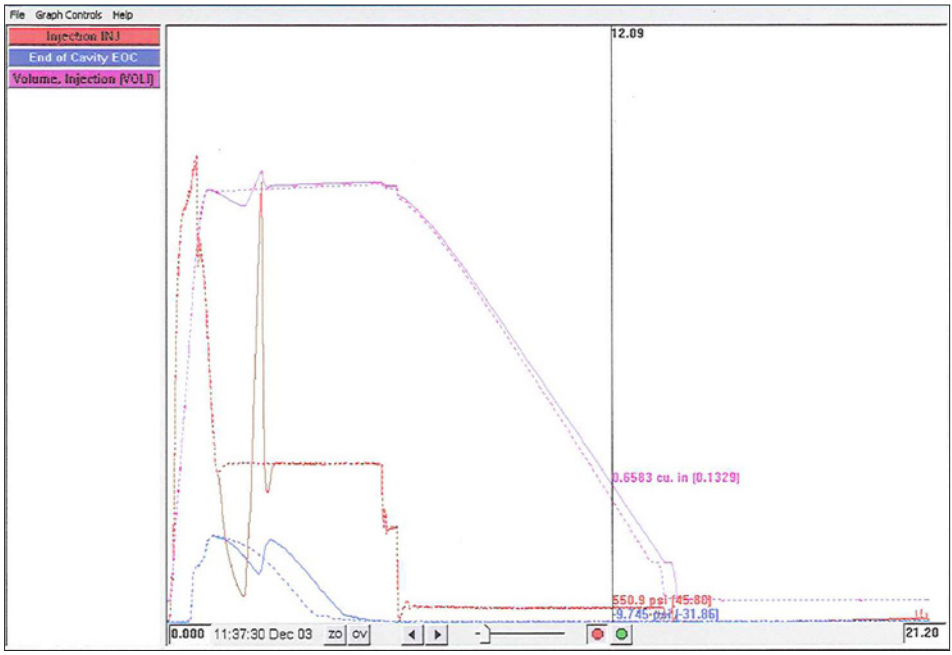


Figure 30.5 Poor velocity to pressure response: note undershoot followed by overshoot in pressure

30.3.2 Flash Troubleshooting Mold Issues

Mold causes may include:

- Parting line damage
- Vent depth
- Trapped plastic
- Support
- Erosion
- Slide deflection
- Mismatch
- Cavity balance

30.3.2.1 Mold: Parting Line Damage

One of the keys to avoiding flash is the requirement of a mold to have a robust shut off between the halves. If the parting line of a mold becomes damaged the shut off between the mold halves will not be sufficient to keep the plastic in the mold.

Damage to parting lines can happen for many reasons including:

- Use of steel tools causing damage
- Excessive removal of steel from polishing or grinding of vents
- Clamping the mold on a part or runner
- Flakes or stringers caught on the parting line
- Erosion from lack of venting

In the case of these conditions, eliminate the root cause of the problem or the flash will return after repair. Further STOP analysis might be required to address the true root cause of the flashing from damaged parting lines.

When a damaged spot is located on the parting line of a mold it must be fixed. The part will have flash at this location until the repair has been completed. Trying to process around parting line damage will lead to a severe restriction of the process capability for a mold.

Laser welding is a very effective means of repairing parting line damage. A skilled laser welder can weld such fine detail that very little work will need to be done to finish polish the parting line detail.

30.3.2.2 Mold: Vent Depth

In most cases more venting is a good thing; however, vents can only be deepened so far based on the material being used. It is vital to determine the maximum vent depth for a given material but it is also important to understand that vent depth is not a "one size fits all" application.

The lower the viscosity of a material the easier it will be for vents to flash. This is important to keep in mind when dealing with materials such as nylon that will experience viscosity variation due to moisture content. Nylons can be very touchy on vent depth because they usually need to be well vented but will flash easily. When needing more venting remember that depth is only one of the parameters of a vent; width can be added to vents without risking flash.

If vents are too deep it will usually be very apparent because the flash will mimic the vent locations and widths. If tabs of flash are located at the vents the vents will need to be addressed.

30.3.2.3 Mold: Trapped Plastic

There are cases when plastic can stick or break and become trapped in a mold. If someone starts up a process without reducing second-stage pressure, it is common for the mold to flash excessively. The excessive flash can often end up stuck in holes, screw heads, and insert lines. This stuck plastic may prevent a complete lockup of the mold because it is acting as a spacer on the parting line.

Always inspect the surface of the mold for any potential trapped plastic. It only takes a small piece of plastic to prevent proper shut off. Many times if there is trapped plastic on the parting line the machine will alarm due to mold safety preventing lockup. Unfortunately, it can be very time consuming to remove all of the plastic that can become trapped after severely flashing a mold. Prevention is the best solution to this problem, and make sure that people start up processes running a fill only shot to help prevent flashing the mold.

30.3.2.4 Mold: Support

When a mold is built there must be steel that provides structure to allow the machine tonnage to apply force from the platen through the mold. Support pillars must be added when mold areas are cleared out. The two common areas that are cleared out and need additional support added are:

1. Ejector box

   The space where the ejector plate travels within the mold must have support pillars added to ensure that the mold does not deflect from cavity pressure (see Figure 30.6; the smaller inner circular shapes are support pillars).

2. Hot runner manifold

   The space in the hot half that is occupied by the hot runner manifold needs to be cleared for locating the manifold. Due to the clearance of the steel around the hot runner manifold support must be added to avoid mold deflection.

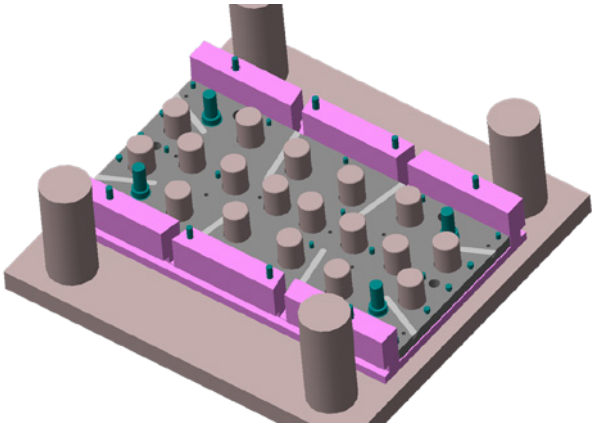


Figure 30.6 Support pillars

If a mold is not adequately supported it will experience plate deflection that will lead to flash. The internal mold deflection will not be measurable from the exterior of the mold like platen deflection is. An internal deflection sensor can be installed in the mold to determine if the mold is actually deflecting.

Support must be designed in before the mold is built and worked into a mold around ejector pins and lifters, so the overall mold design must be considered to achieve optimum results.

Bear in mind that even if a mold has adequate support the support pillars can hob into the mold clamp plate or retainer plates over time. This hobbing of the support plate steel will lead to a lack of support that will cause flash. The mold must be disassembled to check for hobbing of the support, so it is an easily overlooked root cause.

30.3.2.5 Mold: Erosion

Erosion of steel on a mold often occurs in poorly vented areas. If gas is trapped in a location it will eventually start to cause erosion of the steel, which will lead to flash. If steel erosion is detected the location in the mold should be better vented to allow gases to better escape the mold.

30.3.2.6 Mold: Slide Deflection

Any slides that form action outside of the line of draw on a mold must hold forward with enough force to overcome the plastic cavity pressure. If the force on the slide is not adequate the slide will be blown backwards resulting in flash around the detail that the slide is forming.

Hydraulic actuated core pull cylinders must have a large enough cylinder combined with a high enough core pressure to prevent movement of the slide. The calculation for determining cylinder size is simple but relies on an estimate of cavity pressure. To determine the required cylinder size the estimated cavity pressure acting on the area of the core must be compared to the machine core pressure applied to the square area of the cylinder (see the calculation in Figure 30.7).

Hydraulic cores can also be supported with a heel block locking angle that will lock the core in place with steel. In this method the hydraulic core is set before mold close and then the heel block locks the final preload onto the core.

Slides that are driven with horn pins will need a heel block to provide adequate support. The heel block is what will actually preload the slide and resist the force from the cavity pressure. Proper engagement of the heel block is critical to avoiding flash, do not rely on the horn pins to provide the lock for the slide.

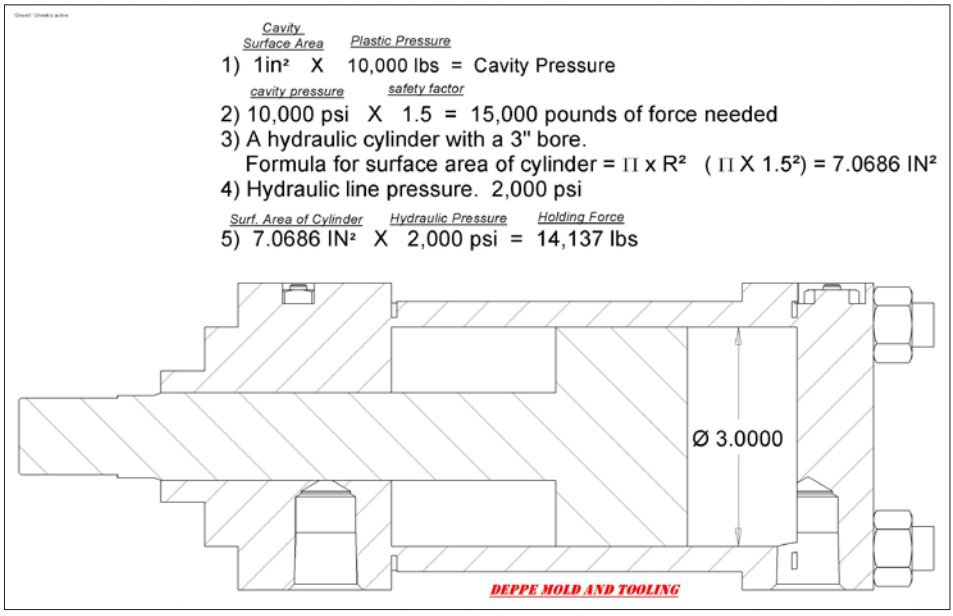

Cavity Surface Area
Plastic Pressure
1) 1in² X 10,000 lbs = Cavity Pressure
cavity pressure
safety factor
2) 10,000 psi X 1.5 = 15,000 pounds of force needed
3) A hydraulic cylinder with a 3" bore.
Formula for surface area of cylinder = Π x R² (Π X 1.5²) = 7.0686 IN²
4) Hydraulic line pressure. 2,000 psi
Surf. Area of Cylinder
Hydraulic Pressure
Holding Force
5) 7.0686 IN² X 2,000 psi = 14,137 lbs
Ø 3.0000
DEPPE MOLD AND TOOLING

Figure 30.7 Calculate required cylinder size

30.3.2.7 Mold: Mismatch

Sometimes what is being called flash is actually a mismatch in the parting line. No matter what is changed on the machine or the process a mismatch will not improve. shows an example of a mismatched parting line. When feeling for parting line flash, true flash can be felt from both directions across the parting line, but a mismatch will only be felt from one direction.

A mismatched parting line can arise from excessive polishing along the parting line. Removing too much steel from one side of the mold during polishing can create a mismatch that is objectionable. Care must be exercised when polishing near a parting line.

Figure 30.8 Mismatched parting line

30.3.2.8 Mold: Cavity Balance

If a multicavity mold is not well balanced a common side effect is that some cavities flash whereas others are under-packed. If all cavities are not balanced for filling and packing it will be very difficult to find a process window that produces acceptable parts in all cavities.

The normal industry standard is that all cavities should be balanced within 3% for a critical part and a maximum of 5% for noncritical parts. The part requirements for dimensional characteristics as well as performance requirements will dictate if the balance should be held to 3% or if 5% is allowable.

Family molds will complicate matters even more because the amount of variation between cavities is driven by the overall size and shape difference between the cavities. Family tools can create a great deal of flash problems due to the inherent cavity imbalance.

For more information on cavity balance see Chapter 12.

30.3.3 Flash Troubleshooting Machine Issues

Some flash causes that are machine related include:

- Tonnage
- Clamp parallelism
- Core pressure
- Mold size
- Toggle wear

30.3.3.1 Machine: Tonnage

Is the molding machine set to the correct tonnage? Is the machine actually achieving the desired tonnage? These are a couple of key questions to ask when experiencing flash. Some machines have tonnage settings that can be adjusted and as such can be set at a too low value for a given mold.

Clamp tonnage requirements are based on many factors including:

- Material type
- Flow length
- Wall thickness
- Cavity pressure
- Gate quantity and location

All of the above factors interact to determine the required clamp tonnage per square inch of projected area. The projected area is the square area of the part that

acts against the clamp tonnage to try to force the mold open. When these factors are considered the simple tonnage per square area calculations can become more complex.

To estimate basic clamp tonnage, calculate the projected area of the part. The projected area of the part should then be multiplied by the tonnage factor of the material, which is usually expressed as tons/square area. Normally material suppliers will provide a tonnage range; for example, ABS may need 3–4 tons/in^2 whereas polycarbonate may require 4–5 tons/in^2. If running a thin wall with high pressures or processes that require high cavity pressure to pack out sink, the required clamp tonnage will be higher. Using an alternate process like gas assist or MuCell® will lead to lower required clamp force.

Another item to watch for is the actual machine tonnage and if that level of tonnage is being maintained. If a machine is experiencing hydraulic leaks in the clamp circuit it may not be capable of maintaining clamp tonnage. Some machines have tonnage settings that allow the clamp to decompress after a time period. Examine the machine for a clamp decompress setting to determine if tonnage is dropping prematurely.

Keep in mind that damage to the machine components can lead to a lack of clamp tonnage. Key areas to verify for damage can include the platen, the tie bars, and the tie bar nuts. A cracked tie bar or tie bar nut will lead to problems with a machine being able to build tonnage in a specific quadrant. A test to determine if the mold or the machine is the root cause is to rotate the position of the mold in the machine, and if the mold is rotated 180° and the flash does not follow along with the rotation the cause is most likely machine related.

Case Study: Tonnage Drop

In this case the machine was only reaching about 95% of the peak tonnage but more importantly it was not maintaining the 95% level. The tonnage dropped during inject forward until it reached a minimum value that was only about 70% of the specified tonnage of the machine. This dropping tonnage led to mold flashing. To resolve the issue a leaking clamp valve had to be replaced. Look for leaking clamp cylinders when diagnosing this type of problem.

A 1000-ton machine that only functions as a 700-ton one can create havoc when trying to establish or troubleshoot a process.

30.3.3.2 Machine: Clamp Parallelism

Has the molding machine been checked on a regular basis to ensure that the platens are square to each other? Has tie bar stretch been verified to ensure that equal

tonnage is being applied? These are questions that could impact the ability of a machine to hold a mold closed. If the clamp unit of the machine is not providing uniformly distributed tonnage to a mold the risk of flash will increase in areas that are not receiving the correct clamp force. Review the machine manual for recommended procedure and schedule for clamp verification.

Tie bar stretch can be determined following the recommended machine specific guidelines. In simple terms conducting a clamp single point adjustment requires the following steps:

1. Verify machine level with a machinist's level. See machine service manual for locations and procedure.
2. Check tie bar nuts to ensure they are tight.
3. Mount single-point frame to machine base.
4. Install four dial indicators on the single-point frame. Each indicator should contact the flat end of one of the tie bars. Adjust all indicators to zero with the clamp open.
5. Close and lock up the clamp on a mold or test block.
6. Check the dial indicators to determine the tie bar stretch for each tie bar. Variation between the tie bars will vary based on tonnage but variation greater than 0.002–0.003 in will call for adjustment.
7. If tie bar stretch is uneven adjust the tie bar nuts to equalize strain on each of the four tie bars. Make sure to follow the machine manufacturer's recommended methods and techniques; this is a simplified reference.
8. Recheck strain and adjust accordingly.

Also general preventative maintenance on platens should account for keeping the platens clean and smooth. Use a large surface area polishing stone to remove any burrs on the platen surface. Wiping platen and mold surface with a cleaner or lubricant such as WD-40 will help to keep surfaces clean.

30.3.3.3 Machine: Core Pressure

To adequately resist plastic pressure against a mold action such as a core the hydraulic pressure of the core circuit must be set properly. If core pressure is set too low the hydraulic cylinder will not be able to maintain a forward position and as a result there will be flash.

Verify that the pressure has been set correctly on the machine controller. It is also important to determine if the machine is actually producing the desired pressure. A hydraulic gauge can be installed in the core circuit to determine the actual amount of hydraulic pressure acting on the core cylinder. Sometimes the pressure relief valve has been set to restrict the core hydraulic pressure regardless of what the set point is on the machine.

Comparison of the cavity pressure acting on the square area of a core can be made with the square area of the cylinder and the applied core pressure. See the calculation in Figure 30.7 for specifics on calculating required pressure and cylinder size.

30.3.3.4 Machine: Mold Size

The obvious problem with mold size is trying to run a mold that is too large for the given machine size. There are cases where even though the mold fits between the tie bars of the press the projected area and cavity pressure required are too high for the clamp to keep the mold closed. It is important to understand that every material has a range of required clamp force expressed in tons/in^2. This range is based on the normal cavity pressures that a material will need to pack out a part. Understand, however, that these are average values that can be affected by things like wall stock, number of gates, flow distance, and dimensional requirements.

The opposite end of the spectrum is when a mold is too small for a given machine size. Normal recommendations are that a mold should cover approximately 2/3 of the platen surface area. If a mold smaller than the recommended minimum is run in a machine, platen wrap can occur. Platen wrap occurs when the oversized platen is literally distorting around the small mold. In a case where platen wrap occurs the clamp force applied near the center of the mold will be reduced, which in turn can lead to flash in the center of the mold.

Always verify what the minimum mold size is for every machine in the shop. This issue not only leads to flash but can also cause mold damage and could also cause machine damage. If a mold is to be run in an oversized press use of outrigger support pillars will eliminate mold damage.

30.3.3.5 Machine: Toggle Wear

Any mechanical item will experience wear over time. Clamp components on molding machines are no exception. As the toggle links and link pins become worn over time it will become more difficult for a machine to achieve a solid clamp lockup. In some cases the toggle becomes so worn that it is visibly sloppy as the clamp locks up. This slop can lead to non-uniform dispersion of clamp forces.

Wear on toggle components should be evaluated over time. A clamp that is running slower than normal or making unusual noises may indicate excessive wear. Clamp rebuilds are time-consuming and expensive but pay off in the long run by reducing mold damage, yielding faster cycle times, and producing better quality parts.

30.3.4 Flash Troubleshooting Material Issues

Material factors to consider are:

- Viscosity decrease
- Moisture content
- Regrind

30.3.4.1 Material: Viscosity Decrease

If the viscosity of a material decreases it will flow easier. The result of this easier-flowing material is that it will fill through thinner gaps, which may result in flash.

All materials will have variation over time. The normal measurement that is available to detect variation in material viscosity is the melt flow index (MFI). Unfortunately, MFI only gives results for very low shear rate (grams/10 minutes), which may not translate well into the molding environment. Some materials will show good correlation between MFI and flow in the mold. Large increases in MFI may indicate a shift in material viscosity that leads to a risk of flash.

The various additives in a material can also impact its viscosity. For example, a decrease in glass percentage could lead to a material that flows easier and potentially leads to flash.

If the flash started immediately after changing to a new box or lot of material, the material is the first place to start investigating. Switch to another lot if one is available to see if the problem persists. If a lot change impacts the issue the root cause may very well be a material issue.

30.3.4.2 Material: Moisture Content

Materials that experience hydrolysis will also experience viscosity shifts as the molecular weight is reduced during hydrolysis. Because hydrolysis is breaking down the chain length of the molecules of the material the result is a material that will flow easier.

When processing wet material, the risk for flash is much increased. This easier-flowing material will flash through parting line gaps that the material typically does not. Always when starting a molding machine and purging out material watch how runny and thin the material is as it leaves the nozzle. Wet degraded material will be very watery and will not maintain any shape of the melt stream in the purge puddle. Also wet material will tend to have a lot of bubbles from the extra gas content. If the melt does not look normal it is a good idea to check out the moisture content before shooting the material in the mold.

For more information on drying see Chapter 9.

30.3.4.3 Material: Regrind

Reground material is normally perfectly fine for reuse in a process, and in fact generation studies out to the fifth generation show that many materials lose very little in their physical properties. The above statement is only true if regrind is handled properly, not contaminated, and either dried or reused immediately. If regrind becomes degraded it will impact the viscosity of the material, which in turn may lead to flash.

Whenever using regrind try to reuse it at the point of generation. To be able to achieve this it is important to design runner systems that are smaller than the allowable regrind percentage. Also do not try to regrind and reuse parts that were degraded. In other words, if a part is scrapped because of splay do not regrind it because all that is going to happen is that the problem on that part will be diluted and spread across many more parts.

Use of regrind is not bad but it must be treated correctly. Keep it clean, dry if necessary, grind to a consistent size, limit dust, and try to reuse it as quickly as possible.

31 Flow Lines

31.1 Description

Flow lines consititute a wide-ranging defect that can appear under a variety of conditions. The typical flow line will appear as a line or loop on the part surface. Also in this chapter on flow lines, the concerns of cold flow and hesitation will be addressed.

Also known as: record grooves, chicken scratches, recording, stalling, hesitation

Mistaken identities: scratches, splay, meld lines, weld lines

31.2 Flow Lines Troubleshooting Chart

Table 31.1 shows the flow lines troubleshooting chart.

Table 31.1 Flow Lines Troubleshooting Chart

Molding Process	Mold	Machine	Material
slow/fast fill velocity	wall stock	velocity to pressure transfer	moisture content
low fill only weight	flow pattern	machine performance	viscosity
melt temperature			filler content
mold temperature			

31.3 Flow Lines Troubleshooting

When troubleshooting flow lines, a short shot analysis can be a significant benefit. To understand the cause of a flow line, the starting point must be determined. There will be cases where a flow line appears to be located with no features around it. Short shots can often show that the initial starting point of the defect was actually further upstream in the flow path.

Often people do not spend enough time looking at short shot progression on parts. Even when conducting a shear viscosity study many people fail to look at the parts but rather just document the process results. Examination of short shots and flow progression will often highlight areas of concern in a mold.

Always eliminate the other possibilities for flow lines; it is frustrating to hear that someone worked on a flow line for hours only to find there was a scratch on the mold surface. Be observant of scratches, color streaks, and splay to ensure that the correct problem is being addressed.

31.3.1 Flow Lines Troubleshooting Molding Process Issues

Process settings that may lead to flow line defects include:

- Fill velocity
- Fill only weight
- Melt temperature
- Mold temperature

31.3.1.1 Molding Process: Fill Velocity

Fill velocity can cause flow-related defects if the velocity is too low or too high. Always verify that the actual fill time and fill only weight match the established process.

A fill velocity that is too slow can lead to hesitation lines in the molded part that indicate where the part struggled to fill. A classic scenario is where what is referred to as record grooves are generated, where the melt takes on a track-like appearance that looks similar to record grooves (see Figure 31.1). A slow fill velocity can also produce meld lines in the flow front that become obvious from a visual standpoint. Try filling the mold faster to see if the flow defects change. During process development observe parts made during filling experiments to determine what fill velocity produces the best results.

Figure 31.1 Flow front showing flow lines (record grooves)

Filling too fast can lead to concerns where the venting of the mold is impacted, a part detail is flowed past and backfilled, or flow front separation occurs, among others. Fast fill rates help with many things in molding including improved cycle times and ability to pack out a part. If troubleshooting flow lines try adjusting the velocity to determine if the problem improves.

31.3.1.2 Molding Process: Fill Only Weight

If a process is transferring early there may be a hesitation in the flow front that creates a line located where the melt flow transferred from first stage to second stage. These lines can be very distinct and are normally located near the end of fill on the mold. Very early transfer positions could result in a line that appears earlier in the molded part.

Verify that the fill only weight is correct. The part should be visually filled to approximately 95–98% fill during first stage. If the fill only weight is light adjust the transfer position to achieve a 95–98% full shot. The fill only weight is a critical part of determining if the mold has been filled consistently. One of the 4M Basic 8 key steps is to verify that the fill only weight matches the documented weight.

31.3.1.3 Molding Process: Melt Temperature

Low melt temperatures can impact the viscosity of the material filling the mold. When running a low melt temperature, the material will be closer to the no-flow temperature for the material and may come close to freezing off during first-stage fill. A too-low melt temperature will also raise the transfer pressure and may in result in a pressure-limited situation.

Check the melt temperature to determine if the material is being processed to the recommended specification. If the melt temperature is running low examine if it is due to set points or actual machine performance. Correct whatever is wrong with

the machine or molding process and recheck the melt temperature. If the melt temperature is running correctly try raising the melt temperature to determine impact on the defect.

Very low melt temperatures will also potentially lead to a record grooving type of defect on the part. These record groove ridges can be a good indicator that the melt temperature of the material should be raised. Record groove defects are often accompanied with a short shot at end of fill.

Low melt temperatures will also impact the quality of meld lines. Always try to determine if the flow line is actually a meld line downstream from the actual location where the material separated. Meld lines can show up literally anywhere from the formation location to the end of the part. Warmer melt temperatures will allow a better-quality meld line.

31.3.1.4 Molding Process: Mold Temperature

Cold mold surface temperatures can lead to a variety of flow-related defects. Long flow lengths in cold molds can lead to hesitation due to the material freezing excessively. This impact is compounded by flow length, thinner wall stock, and first-stage velocity. Thin wall stocks with long flows will experience more flow limitations and slow fills will see a greater impact of the frozen layer of material that has contacted the mold.

Check the process to determine if the mold temperature is set correctly, and make the required adjustments if it does not meet specification. Watch the defect; if it only occurs on startup parts it may be an indication that mold temperature needs to be raised. Remember that prior to injecting plastic the mold surface will be very close to the incoming water temperature but will increase as plastic is injected into the cavity. This increase may be slight but it still may point to a direction in troubleshooting.

31.3.2 Flow Lines Troubleshooting Mold Issues

Mold-related issues can include:

- Wall stock
- Flow pattern

31.3.2.1 Mold: Wall Stock

Look for areas in the molded part where the wall stock chokes off the flow front. Such areas often will result in a flow line from the melt hesitating as it hits the restriction. Some thin sections will end up being backfilled well after the flow front has progressed past the location of the thin area. These backfilled locations can

have hesitation lines or even a blush-like appearance from the melt being forced through the thin wall.

Always try to maintain uniform nominal wall stock. There have been cases where even a 0.002–0.003 in difference in wall stock created a varying flow front that led to a flow line. Short shot studies can help detect these types of problems. If the wall stock must vary try to blend the surrounding wall stock to allow a better transition into the thin section. Identify wall stock variation and be prepared to make process adjustments to accommodate the design issues, but note that this will reduce the process window for the mold. Run a flow analysis on the design prior to cutting steel to look for areas of backfill and potential hesitation or non-filled areas. Making changes on the CAD model is much more cost effective than adjusting steel after the mold has been produced.

There may be cases where a flow leader or locally thickened area is needed on a part to get adequate flow throughout a mold. Again this can be modeled in during flow analysis to optimize the design prior to cutting steel. Remember when adding flow leaders that they should be blended whenever possible to minimize read-through on the class "A" surface.

31.3.2.2 Mold: Flow Pattern

There are cases when the flow pattern in the mold leads to flow lines. The location of the gate determines how the mold will fill; it should be located to provide optimized flow. If the flow length is too far from the gate the flow front will eventually start to stall. A stalled flow front will lead to either a hesitation line or record groove type of defect.

In a mold that is sequentially valve gated the downstream gates may surge into the cavity when opened leading to flow defects on the part surface. If flow defects are appearing when sequentially gating try adjusting the timing of the gate actuation to determine if the defects are improved. Hot runner suppliers now offer modulation of valve pin actuation that can offer some potential improvements of downstream gate defects.

In molds that violate nominal wall stock guidelines the flow pattern through the varying wall stocks may yield meld lines that are called out as flow lines. These meld lines will appear in the direction of flow and can be very subtle. Short shoot the mold through the areas of concern to determine if a meld is occurring during fill.

Figure 31.2 shows an example of a short shot showing an unbalanced flow progression which resulted in a meld line forming. This meld line was scrapped out as a flow line, but it was only after running short shots that the true cause was understood. Using STOP methodology, it was possible to systematically work through this problem until the root cause was identified.

Figure 31.2 Uneven flow progression resulting in a meld line that was called out as a flow line

Computer flow analysis will help identify areas that will be prone to experiencing flow pattern issues. Running an analysis before a mold is built will help minimize the risk of fill pattern problems. If the mold has already been built flow leaders or flow restrictions may need to be added to compensate for the flow issues.

31.3.3 Flow Lines Troubleshooting Machine Issues

Machine impacts on flow line defects include:

- Velocity to pressure (V to P) transfer
- Machine performance

31.3.3.1 Machine: Velocity to Pressure Transfer

If the machine is not doing a good job when transferring from first-stage velocity control to second-stage pressure control a witness line may appear on the part. This witness line can be considered a flow line because it actually indicates a spot where the plastic flow hesitated. If the machine pressure is dropping significantly during transfer the melt can hesitate and impact both the cosmetics of the part surface as well as the ability to effectively pack out the part.

An example of a poor velocity to pressure response is shown in Figure 31.3. Notice in this example that the machine pressure drops to zero, the screw bounces back severely, and the cavity pressure drops to zero. This example created a distinct hesitation just prior to complete cavity fill, which resulted strange flow/hesitation lines near the end of fill for this part.

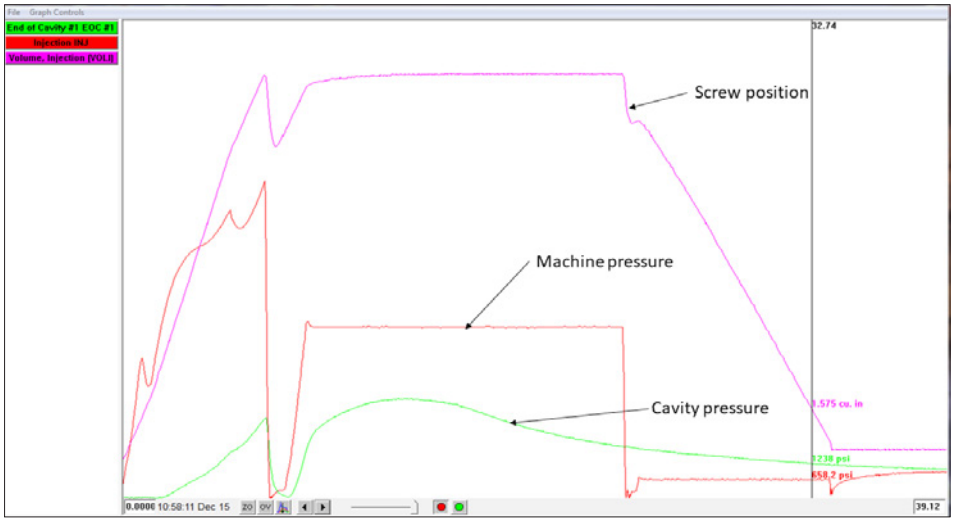


Figure 31.3 Poor velocity to pressure response

Poor velocity to pressure response can be caused by several factors including:

1. Transferring too early

   Normally the transfer from first stage to second stage occurs around 95–98% full. Transferring too early will create a lag in how the machine can build pressure. Frequently early transfer will show a dip in pressure followed by a pressure overshoot as the machine tries to reach the set pressure.

2. Incorrect second-stage velocity or volume setting

   If the volume setting for second-stage pressure is set too low the machine will lag as it tries to build pressure. Adjusting the second-stage volume setting can

often improve this problem. Remember that on machines with both a pack and hold volume both may impact the process.

3. Worn or improperly tuned hydraulic system

   Pressure can only build as fast as the machine can supply flow, so if the pump output is lacking the machine will struggle to achieve a quick response.

   For more on velocity to pressure response see Chapter 8 on machine performance.

31.3.3.2 Machine: Machine Performance

If the machine cannot achieve the set points of a process the result is frequently a part defect. Evaluate the machine to determine that it is achieving the documented process.

For details see Chapter 8 on machine performance.

31.3.4 Flow Lines Troubleshooting Material Issues

Some key items to consider from the material side include:

- Moisture content
- Viscosity
- Filler content

31.3.4.1 Material: Moisture Content

Some materials will experience viscosity variation due to moisture content. Nylon as an example can be molded over a wide range of moisture content (0.05–0.3% or more). Molding nylon across a wide range of moisture content will lead to noticeable changes in viscosity with nylon of higher moisture content flowing much easier (lower viscosity). Due to these moisture-induced viscosity changes a variety of flow line type of defects may appear. If the material becomes excessively dry the viscosity may actually become high enough to lead to a pressure-limited situation, which will in turn result in fill time variation.

Verify that the material being molded is adequately dried. Also examine how much the moisture content is varying over time. If the process is seeing large swings in moisture content many odd flow defects may show on the part.

See Chapter 9 on drying for more information.

31.3.4.2 Material: Viscosity

All materials will have a natural variation in viscosity lot to lot. This variation can be compounded when running wide-specification materials and with the addition of regrind. If a defect shows up immediately after changing to a new lot of material, try switching lots again to see if the defect improves. Keep in mind that the change in material may also be linked to moisture content and not a true material problem.

If running with regrind try running with 100% virgin material. Regrind can in cases raise or lower the viscosity of the material and may create flow-related defects. Viscosity can increase with regrind use because some of the lubrication package may have been degraded during the original molding process. Viscosity may drop because the material experienced a level of breakdown during processing, resulting in a smaller average molecular weight of the molecules. Do not assume that regrind will always have the same impact with all materials and processes.

31.3.4.3 Material: Filler Content

In general terms, the higher the level of fillers added to the plastic the more likely flow defects will be seen. During the design and specification stage of the project, evaluate if a 15% glass-filled material will work instead of a 30% glass-filled material. Higher filler contents will create a higher-viscosity material resulting in more challenges filling a given mold. Also when using filled materials the fillers may come to the surface of the part and highlight flow issues. Metallic flakes are notorious for showing distinct flow lines in complex geometries (see Figure 31.4).

Verify that the correct filler content is being used for a given material. If using a colorant additive with filler content ensure that the correct letdown ratio is being used. When using a volumetric additive feeder, the letdown ratio is based on a formula for a given auger size, so make sure that the correct auger size is being used for the job that is running.

Figure 31.4 Flow lines with metallic-filled concentrate

32 Glass Fibers on Surface

32.1 Description

When molding with glass-filled materials a common issue is that the surface of the part will display uneven appearance from the glass fibers. Figure 32.1 shows a magnified view of glass fibers on the surface of a part.

Also known as: glassy, dull, gloss variation

Mistaken identity: splay

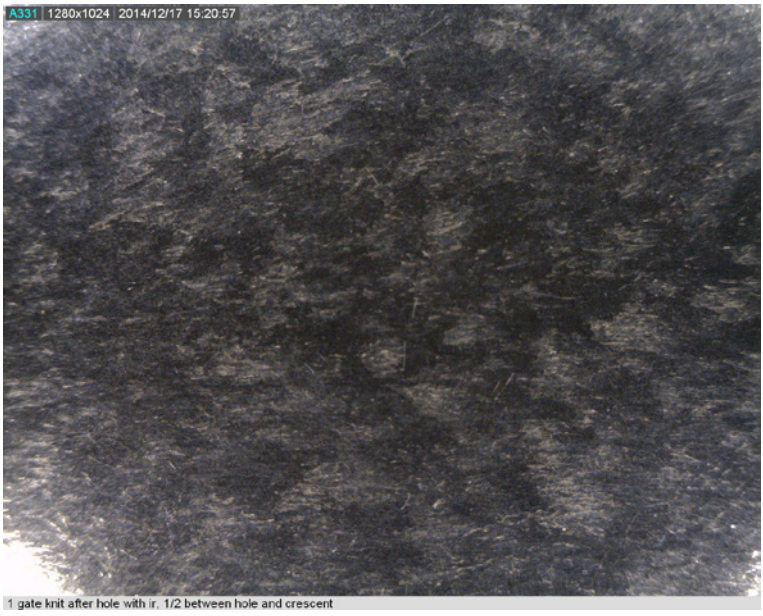


Figure 32.1 Glassy part surface

32.2 Glass Fibers on Surface Troubleshooting Chart

Table 32.1 shows the glass fibers on surface troubleshooting chart.

Table 32.1 Glass Fibers on Surface Troubleshooting Chart

Molding Process	Mold	Machine	Material
slow fill velocity	lack of venting	machine capability	moisture content
low melt temperature	hot runner temperatures	machine performance	viscosity
low mold temperature	flow length		glass content
	gating		

32.3 Glass Fibers on Surface Troubleshooting

32.3.1 Glass Fibers on Surface Troubleshooting Molding Process Issues

The molding process can have a large impact on glass showing on the surface of a part. Some of the process factors that influence cosmetics of a part include:

- Slow fill velocity
- Low melt temperature
- Low mold temperature

32.3.1.1 Molding Process: Slow Fill Velocity

A slow fill velocity will tend to allow the glass fibers to come to the surface of the part. Normally the faster the mold is filled the better the surface will appear. A fast fill creates a resin-rich surface that will help keep the surface looking uniform and not glassy.

If glass on the part surface is a problem verify that the fill time and fill only weight of the part match the documented process setup sheet. Transferring early can lead to end of fill locations being more glassy-looking then the rest of the part.

If the fill velocity is not optimized try filling faster to determine the impact on the glassy appearance. Make sure that the machine is not running at a point where it has become pressure limited (fill pressure greater than 90% of maximum available

machine pressure). If the machine is running close to the pressure limit the fill velocity is likely to vary over time and the appearance of the part will also.

If the mold is being run in a machine other than the one it was sampled in make sure to evaluate if the machine is capable of achieving the required fill velocity throughout the shot. If the fill time cannot be obtained by increasing fill velocity investigate if the machine can and does achieve the velocity required. See Chapter 8 on machine testing for more on evaluating velocity performance.

One key to filling fast enough with glass-filled materials is adequate venting. If the mold is not vented well enough it will not be possible to achieve adequate fill velocity without burning (see Section 32.3.2.1).

32.3.1.2 Molding Process: Low Melt Temperature

High melt temperature will help get a resin-rich surface for the molded part. If the melt temperature is too low the part will tend to look glassy. Also if melt temperature is low the fill pressure may increase and cause the process to become pressure limited resulting in slowing fill velocity.

A melt temperature verification will determine if the process is being run at the proper temperature. Do not evaluate melt temperature based on barrel temperature set points. The melt temperature must be measured either with a melt probe or an IR thermal device. Verify against the documented setup sheet and the recommended melt temperature of the material supplier.

Raising the melt temperature may allow the mold to fill with better surface appearance. If the part surface is too glassy in appearance try to increase the melt temperature toward the upper end of the recommended process window. Avoid increasing the melt temperature too much or the material may degrade and cause other defects.

Be conscious of using high back pressure when molding glass-filled material. As the fibers travel down the screw, excess shear will cause the fibers to break reducing the length of the fibers. With long glass fibers this becomes even more critical to the performance of the plastic in the molded part. The thing to beware of is that the shorter glass fiber lengths will look better but will lose physical properties.

32.3.1.3 Molding Process: Low Mold Temperature

Low mold temperatures are a leading cause of glass on the part surface. To keep glass fibers buried, the mold temperature often needs to be run at the upper limits of the recommended temperature.

In many cases to get optimized surface appearance a high-temperature thermolator must be used to allow the mold to be run at temperatures above 212 °F. If running high temperatures extra caution should be utilized to protect people from

hazards like cooling line ruptures. High-temperature lines are a critical factor when running with high mold temperatures. Do not try to cut costs on this. Hot oil is another option for achieving even higher mold temperatures.

Clearly label all of the thermolators in the shop. This way if a mold needs a high-temperature thermolator it will be clear to everyone and when a replacement thermolator is swapped for repair the correct replacement will be used. If someone swaps a standard thermolator for a high-heat unit it will not be possible to achieve the mold temperature needed to keep the surface looking good.

32.3.2 Glass Fibers on Surface Troubleshooting Mold Issues

A few mold-related issues will potentially impact the amount of glass fibers showing on a part surface such as:

- Lack of venting
- Hot runner temperatures
- Flow length
- Gating

32.3.2.1 Mold: Venting

Lack of venting in a mold will cause the flow front velocity to slow as the mold is filled. If the mold is not adequately vented the gas will be trapped ahead of the melt flow stream and act as resistance to the plastic filling the mold.

The other key constraint from lack of venting is the possibility of burning occurring at the faster fill speeds that for glass-filled materials provide the best cosmetics. If the mold cannot be filled at an optimized velocity due to lack of venting the mold must be optimized for the process. Trying to process around a lack of venting will at best produce a part that is borderline on quality and will be a constant problem in manufacturing.

Knit lines can be especially challenging with glass-filled materials and the appearance at the end knit line or end of fill areas will tend to look worse than the rest of the part. To combat this there will be cases where the end of fill knit line location actually needs an overflow to allow the material to have wide open venting and create more of a meld line than a straight butt weld line.

Case Study: Glass-Filled Venting

In this particular part molded with 33% glass-filled nylon 6, the process was optimized and providing good surface appearance everywhere but the end of fill knit lines (two locations). A sample was run with one of the two gates

blocked off and the part filled fine; however, the end of fill knit line was still visually unacceptable. Because the new end of fill location ended up at the location of the second original gate a decision was made to try blocking the runner to the second gate but leaving the gate and a small section of the runner open. This open gate and runner section allowed the material at the end of fill to overflow from the part and created a visually acceptable knit line. The wide-open gate allowed plenty of venting at this knit line and allowed the material to flow in a meld line fashion.

32.3.2.2 Mold: Hot Runner Temperatures

If hot runner temperatures are run too low the material might be cooling off as it sits in the hot runner manifold. Hot runner temperatures should be set at the melt temperature of the material to avoid allowing the material to heat or cool when it is in the manifold.

Temperatures in a hot runner system should be controlled with a thermocouple rather than with an amperage percent setting. Good thermal control of the hot runner system will allow for maintaining an optimized melt all the way through the melt delivery system. Remember that a hot runner mold is just an extension of the melt delivery system.

32.3.2.3 Mold: Flow Length

Long flow lengths with glass-filled materials can often lead to situations where the machine is either pressure limited or close to it. When the machine is pressure limited it will not maintain volumetric flow rate, which gets worse the further the mold fills. As the flow rate decreases glass on the surface will tend to be more evident. The main task concerning flow length is to evaluate number and location of gates during design. Verify with the material supplier recommended flow length to wall stock ratios.

32.3.2.4 Mold: Gating

As mentioned in Section 32.3.2.3 the number and location of gates are critical to allowing adequate filling of a mold. It is also important to watch for excessive restrictions in gates or hot drops that lead to large increases in fill pressure. If the gate is too restrictive the process may become pressure limited, which will in turn lead to a decreased fill velocity capability. Watch machine pressure curves during fill to see if there is a very rapid rise at start of fill because this may indicate that there is a restriction that is driving the pressure high and potentially closing the process window.

32.3.3 Glass Fibers on Surface Troubleshooting Machine Issues

The key machine-related impacts on glass fibers on the surface include:

- Machine capability
- Machine performance

32.3.3.1 Machine: Machine Capability

The first thing that must be adequate on a molding machine is the injection velocity. To achieve an attractive part a molding machine must be able to inject fast enough to provide a resin-rich surface. If the machine cannot reach a fast-enough fill velocity the part will have a glassy surface. See Figure 32.2 for an example of poor velocity linearity between set velocity and actual velocity.

Velocity setpoint	Shot size	Decomp	Transfer position	Fill time	Velocity actual	% difference
0.25	3.3	0	1.2	9.86	0.21	14.8%
0.5	3.3	0	1.2	4.7	0.45	10.6%
1	3.3	0	1.2	2.95	0.71	28.8%
1.5	3.3	0	1.2	2.36	0.89	40.7%
2	3.3	0	1.2	2.03	1.03	48.3%
3	3.3	0	1.2	1.64	1.28	57.3%
					Range	46.7%

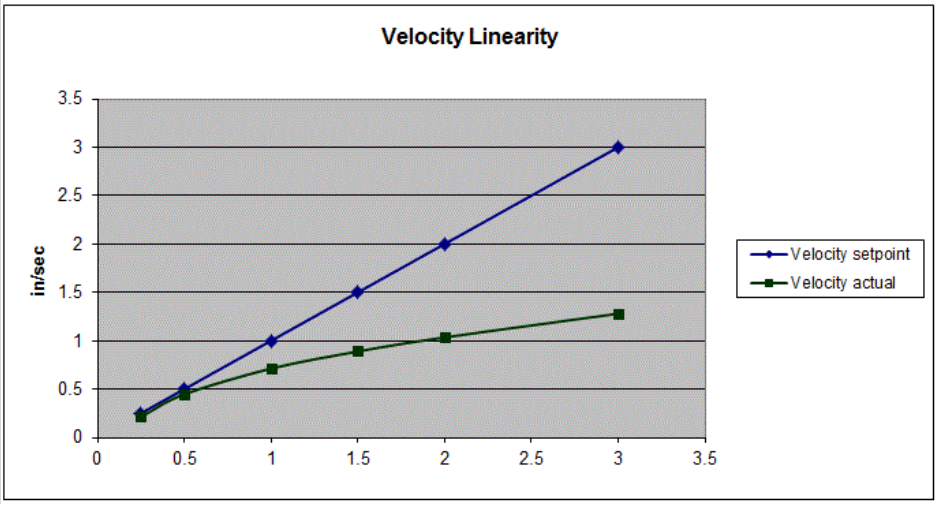


Figure 32.2 Velocity linearity form: notice poor linearity

The second key to a machine being adequate for processing glass-filled material is the pressure capability. It does not matter how high the velocity can be set if the machine does not have enough pressure available to achieve the set point. High-pressure molding machines allow faster fill rates for longer flow lengths.

Case Study: Machine Capability Causing Glassy Surface

In this case the process for a 33% glass-filled part was developed with a different machine than its home production machine. Both machines had a controller that had a velocity set point of 6 in/sec. The original machine was able to achieve the 6 in/sec but the home machine could only achieve just over 5 in/sec. At the 6 in/sec fill velocity the parts looked cosmetically good but at 5 in/sec the parts had a glassy surface. Velocity investigations showed that the home machine just flattened out above 5 in/sec; no matter what number was entered the machine did not get there.

This speaks to processing to the lowest process capability of any machine that the mold will run in. If the process is developed in a machine that is more capable the process cannot be repeated in the less capable machine. This is another good reason to test molding machines (see Chapter 8 on machine performance).

32.3.3.2 Machine: Machine Performance

See Chapter 8 on machine performance.

32.3.4 Glass Fibers on Surface Troubleshooting Material Issues

Material concerns may include:

- Moisture content
- Viscosity increase
- Glass content

32.3.4.1 Material: Moisture Content

There will be cases where the moisture content of a material has a large impact on the surface quality of the molded part relative to glass fibers. This is especially a problem when dealing with nylons or polyesters because both of these materials experience high levels of hydrolysis when processed with moisture present. If a material is run on the higher side of moisture content the viscosity of the material will be reduced, which translates to an easier-flowing material that will help bury the glass fibers better.

An area to be especially aware of is when processes are developed in summer weather conditions versus winter conditions. During the summer high humidity will tend to cause moisture content to run higher than it will in the winter. If a nylon process is developed during the summer months with moisture on the high end of the specification (0.2%, for example) the viscosity of the material is going to

be lower due to hydrolysis degradation of the material. This material will give a much better looking surface than will material at the other end of the spectrum (0.01% moisture content, for example). Keep this in mind when developing processes in the summer. It is often a good idea to allow for some extra drying time to help drive down the moisture content.

One of the challenges in injection molding is dealing with environmental factors. It is nice to have a climate-controlled molding operation but many shops do not have this luxury. Because summer climate versus winter climate can have a large impact on a molding process it puts greater importance on doing an adequate job during process development to try to accommodate potential seasonal variation. This accounts for why there is a large range of drying times for nylon: saturated summer material may require 12 hours to drive the moisture percentage to an equivalent of what 2 hours does in the dry winter months.

Further information on drying can be found in Chapter 9.

32.3.4.2 Material: Viscosity Increase

If the incoming material is varying in viscosity from lot to lot the ability of the material to flow well may be compromised. If running a material that is considered to be "wide spec" there will be a much greater likelihood that the material will experience enough of a shift in viscosity that the process will be impacted. "Wide spec" material by its very nature is going to have greater variation from lot to lot.

If the issue seems to correspond with a lot change in material, it is a good idea to try an additional lot of material to help determine if there is truly a problem. Occasionally there will be processes that have very good correlation in molding window with melt flow index (MFI).

32.3.4.3 Material: Glass Content

If the incoming material is experiencing variation in glass content there may be variations in the appearance of the part surface. As the percent content of glass increases the likelihood of seeing glass fibers on the part surface will also increase. Also if the glass percentage increases so will the viscosity of the material, which may lead to concerns discussed in Section 32.3.4.2.

There are other factors involved in the use of glass such as the sizing agent that allows the glass to be wetted out by the polymer. These sizing agents often have a silane component that helps the plastic to bond with the glass fiber. If the glass fibers are not being encapsulated or wetted out by the polymer they will tend to be more visible on the part surface.

Case Study: Glass Fibers

In this case a 33% glass-filled nylon 6 was being molded for various cosmetic parts. The material was processing well and the parts were being produced without concerns. When a new lot of material showed up and was introduced into the molding machine an immediate shift in surface quality was detected. The parts now had a very glassy surface and looked terrible. Checks were conducted on all of the 4M's as well as moisture content but nothing was found to be out of line.

As the material was introduced into additional molds the same problem was experienced.

The material supplier was contacted to review the problem. The material supplier was very transparent and communicated that on this given lot of material they had used an alternate source for the glass fibers. Investigation showed that the alternate glass fibers had an issue with the sizing agent that prevented the nylon from wetting out the fibers.

A new lot of material was quickly shipped in and the problem was gone. All material from the bad lot was returned to the supplier.

It is also important that design engineers are given feedback on the differences in processing as the glass percentage increases. If there is no good reason to go with a higher glass content the processor will pay the price at the end of the line. Sometimes engineers will want to swap to a higher glass percentage or from short glass to long glass to get a stronger part. They must understand that changes like this do not come without a cost, and the cost is often part cosmetics.

33 Gloss Variation

33.1 Description

Variations in gloss on a part will lead to cosmetically unacceptable parts. If the part has areas that are glossy and areas that are dull the variation in appearance can be very unattractive. Figure 33.1 shows an example of gloss variation.

Also known as: spotty, glossy, dull

Mistaken identities: sink, scuff

Figure 33.1 Gloss variation

33.2 Gloss Variation Troubleshooting Chart

Table 33.1 shows the gloss variation troubleshooting chart.

Table 33.1 Gloss Variation Troubleshooting Chart

Molding Process	Mold	Machine	Material
second-stage pressure	mold surface finish	machine performance	material type
second-stage time	cooling		additives
fill velocity	venting		
fill only weight	inconsistent wall stock		
mold temperature			
melt temperature			

33.3 Gloss Variation Troubleshooting

Gloss is determined by the nature of the surface of the mold, i.e. polished or grained, and how well the plastic replicates the mold surface. The first step of troubleshooting gloss problems is to establish if the mold surface is being replicated. As an example, if a mold surface is a sand-blast finish the part will be dull no matter what the process is (see Chapter 6 for more on mold finish).

Another item to keep in mind when reviewing gloss callouts is that parts that are on the two ends of the gloss spectrum will tend to show scratch and mar defects easier than parts with a more mid-range of gloss. Sometimes the difference between a 2.5 and 3.5 gloss reading can make a major difference in the appearance of the part and the ability of the part to look good after handling. Extremely low gloss levels such as a 2.5 can make it very challenging to get a good-looking part.

It is important to understand that in many cases gloss problems are actually truly a read-through (difference in gloss due to differential pack and shrink at wall stock transitions).

33.3.1 Gloss Variation Troubleshooting Molding Process Issues

Common molding process related problems include:

- Second-stage pressure
- Second-stage time

- Fill velocity
- Fill only weight
- Mold temperature
- Melt temperature

33.3.1.1 Molding Process: Second-Stage Pressure

The most important process setting for gloss on molded parts is the pressure applied to the plastic in the cavity. If not enough pressure is applied to the plastic the molded part will not replicate the finish of the mold well. This applies for polished surfaces, textures, and sand-blasted finishes. Think about the surface of the molded part compared to the mold surface: is the plastic being packed into all of the microscopic surface detail of the surface of the mold?

Examining mold surfaces under magnification will show a series of polish scratches on polished surfaces and a series of peaks and valleys on a textured part. To have adequate replication of these microscopic surface details the plastic has to be pressurized at the appropriate levels. If the pressure is not high enough, the surface will not reflect the details of the mold surface.

Variation in pressure across a mold will result in a potential variation in gloss levels across the part. Low-pressure areas will not replicate the surface as well, and will typically have a different gloss appearance when compared to better-pressurized areas. This can often manifest as a distinct difference between areas near and farthest from the gate. Also watch for areas that fill quickly and freeze before being packed out.

One of the challenges faced in injection molding is trying to minimize the pressure drop across the mold cavity. Some of the keys to minimizing pressure drop include filling fast to avoid viscosity variations during first-stage injection. Proper gate placement is also critical to ensuring a minimized pressure drop. To determine gate quantity and location a flow analysis software such as Moldflow® is a valuable first step before steel is cut.

When troubleshooting for gloss problems check that the second-stage pressure is set at the correct value (do not forget to account for intensification ratio). Also check to see if the correct nozzle and nozzle tip are being used because a mixing nozzle will result in a large pressure drop between the machine and the end of the cavity.

Cavity pressure transducers will provide very useful data on the actual pressure within the mold. If a transducer is located near the gate and at the end of fill location, accurate data on what is occurring in the cavity will be easy to obtain.

33.3.1.2 Molding Process: Second-Stage Time

Second-stage time that is not long enough to ensure gate seal can lead to a glossy area near the gate. If the plastic is not retained in the cavity until gate seal there will be a depressurization of the plastic near the gate. The release of plastic near the gate will create a localized low-pressure area that will typically have a different gloss level than the rest of the part.

Verify that the second-stage time is set correctly per the documented process. A gate seal study should be conducted during process development to determine the appropriate second-stage time. If a gate seal study was not conducted take the time to complete a gate seal study to understand the correct time required for establishing gate seal.

33.3.1.3 Molding Process: Fill Velocity

Generally, a faster fill velocity will provide less pressure drop and yield better surface replication. Faster fill rates tend to provide a glossier surface whereas surfaces obtained with slow fill rates tend to be duller. Faster fill velocities also lead to a part that will pack out more uniformly.

Verify that the fill time is correct according to the documented process. Fill velocity settings may not match, but fill time should. If fill time and fill only weight replicate the desired process that ensures that the same volumetric flow rate occurred.

33.3.1.4 Molding Process: Fill Only Weight

If the fill only (95–98%) shot is too light due to an early transfer there may be a gloss difference that forms in a sharp line where transfer occurred. An early transfer can create a very distinct area at end of fill that looks different.

Check the fill only weight to ensure that the part is filled to 95–98% during first stage. If the shot is short the transfer position should be adjusted to provide an adequate fill shot. Be aware that filling too short can lead to a lag in second-stage pressurization. Figure 33.2 shows process monitoring data on a process with an early transfer. Note how the screw travel curve moves significantly after transfer. As the screw continues to move forward the machine is trying to reach the set second-stage pressure but it takes several seconds. Early transfer creates many issues but gloss variation is an often-overlooked problem.

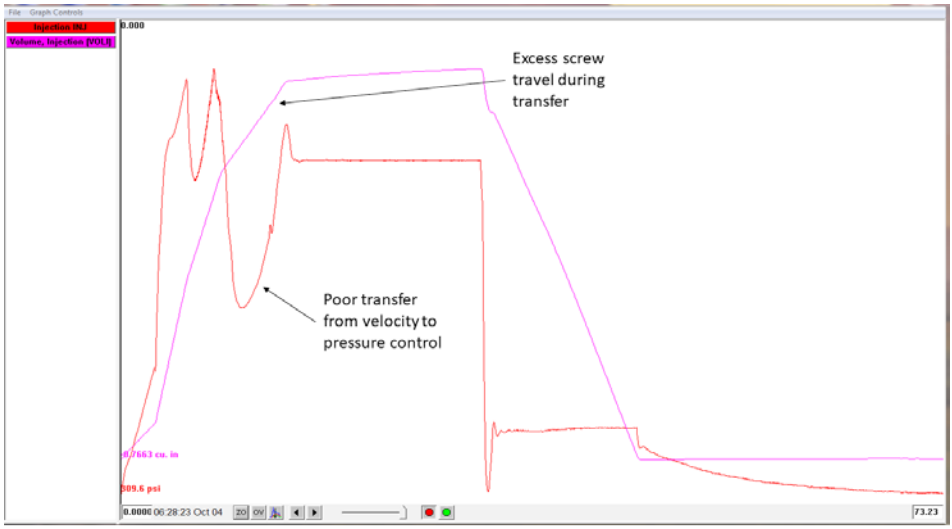


Figure 33.2 Process monitoring curves showing an early transfer

33.3.1.5 Molding Process: Mold Temperature

A low mold temperature can tend to produce a lower gloss level whereas a higher mold temperature can produce a higher gloss level. However, this is another factor that impacts how well the plastic replicates the mold surface, and hotter mold surfaces lead to better replication of the mold surface.

In recent years rapid mold heating and cooling systems have been used to improve the replication of the mold surface. The extra heat that is applied to the mold surface allows the plastic to pack out against the steel much better. Use of rapid mold heating and cooling can yield very high-gloss surfaces when molded on a polished surface. If the mold has a micro texture the replicated surface can be very low in gloss.

There are times when high mold temperatures lead to a rippled surface that will show variation in gloss. The heat of the mold creates an effect where the wall is collapsing away from the mold surface that is sometimes referred to as “heat sink”. Reducing the mold temperature or adding second-stage pressure will often resolve this defect.

33.3.1.6 Molding Process: Melt Temperature

When molding with a low melt temperature the pressure drop across the mold will be increased. With the increase in pressure drop many times a variation in gloss will occur. If the mold is not pressurized at a uniform level there is likely going to be variation in gloss levels.

Verify that the melt temperature matches the documented process. If the melt temperature is wrong then evaluate the following:

- Barrel temperatures set point versus actual
- Back pressure
- Screw recovery rate

33.3.2 Gloss Variation Troubleshooting Mold Issues

One of the biggest impacts on gloss levels of a molded part is the mold itself. The main factors that impact the gloss level are:

- Mold surface finish/texture
- Cooling
- Venting
- Inconsistent wall stock

33.3.2.1 Mold: Surface Finish/Texture

One of the biggest contributors to the gloss of a molded part is the mold surface finish. A sand-blasted mold surface will not produce a high-gloss piano black gloss level no matter what is done to the process.

There is a wide range of finishes that can be used for an injection mold. They include:

- Polish
- Texture
- Sand/glass blast

All three of the above options also have many different levels that can impact the part appearance. For example, a polished surface can be polished to a variety of gloss finishes.

If nonuniform gloss is a problem on a part, the surface of the mold should be inspected for areas of variation on the surface. To effectively do this a strong light is often needed to avoid shadows on the surface. Many times upon inspecting the mold surface, areas of worn texture or buildup will be visible. Sometimes a thorough cleaning of the mold surface will eliminate the variation and other times the mold will need to be sand blasted. Inspecting the mold surface can eliminate a lot of troubleshooting time because if the gloss problem is on the mold surface, processing will not resolve it. Buildup can also be an issue on polished surfaces (see Figure 33.3).

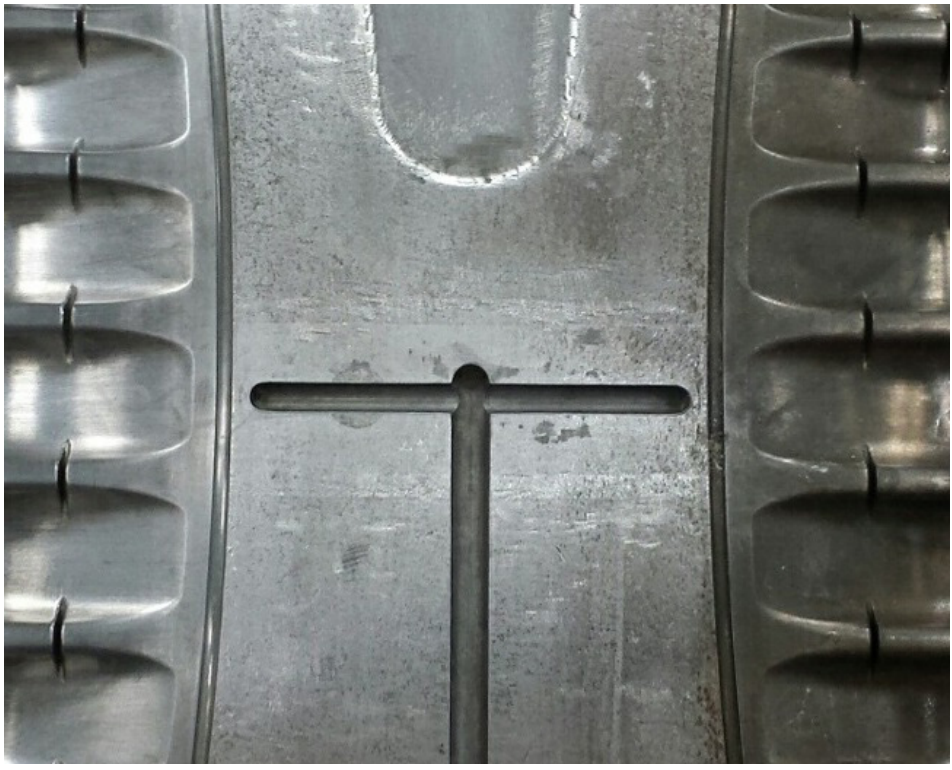


Figure 33.3 Mold surface buildup (right) versus clean surface (left)

Case Study: Polish Buildup

In this case the material being molded was a wood-filled polypropylene. The mold had a polish level of approximately SPI A3, which produced nice glossy black parts when molded with straight polypropylene. An extended run was conducted for a special order of wood-filled polypropylene parts that molded well; however, when the material was changed over to the black polypropylene the gloss level of the parts was dull and inconsistent. Examination of the mold surface showed there was a large amount of buildup on the mold surface and the steel had even tarnished from the gases from the wood-filled material. After many hours of scrubbing with polishing paste, the mold was finally ready for running at high gloss.

Often when a mold texture is used there will either be a secondary micro texture or a sand-blasted finish that creates additional small details that the plastic must be packed into. Under magnification these peaks and valleys will be easy to see and examine for detail. In Figure 33.4 texture details can be viewed under 200× magnification: notice the peaks and valleys that the material must replicate.

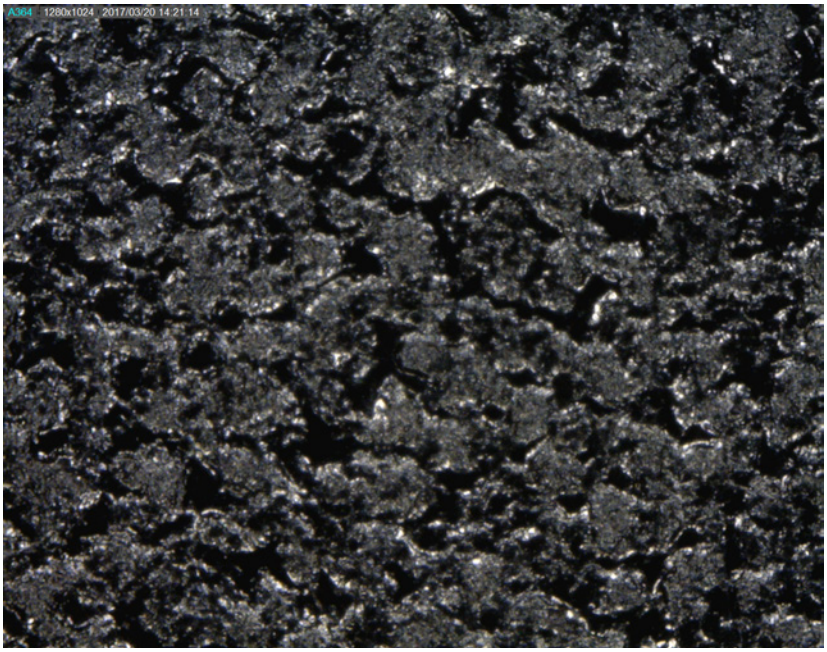


Figure 33.4 Molded part texture under magnification (200×)

Even molds that have a high-polish finish such as a SPI A1 will have microscopic polish lines. As the mold is polished to a given level, finer and finer grits will be used with each grit level erasing the previous polish lines. Under magnification the polish lines are apparent even though the mold and part may have a mirror appearance.

33.3.2.2 Mold: Inconsistent Wall Stock

When there are wall stock changes in molded parts there will often be a gloss difference between the areas of different wall stocks. Due to the different wall stocks the plastic will experience variation in pack and cooling rates. If the plastic does not experience the same pack and cooling conditions throughout the part due to wall stock variation, gloss differences can be expected. Note that this gloss change is frequently called read-through.

Whenever a wall stock transition must occur try to blend the transition over an area rather than having an abrupt change from one thickness to the next. Transitioning wall stock over an area will allow any gloss variations to be less apparent.

33.3.2.3 Mold: Cooling

Gloss levels are impacted by mold temperature as was discussed in Section 33.3.1.5. If the water layout for the mold does not provide uniform cooling the re-

sult can be variation in gloss levels on the molded part. Efficient cooling of a mold is critical to final part quality and must be focused on during mold design.

If gloss differential is occurring in specific areas on the part, review the mold design to determine if there is adequate cooling in this location. Thermal images of the parts will help to highlight areas of poor cooling. Surface probes can also be used to determine the actual mold steel temperature and look for heat concentrations.

If an area on a mold is hotter than the rest of the mold verify why; it may be due to a lack of water near the area, a plugged water line, or a mold that was incorrectly watered. An important part of molding setup is ensuring that the cooling for the mold is consistent every run. Water diagrams must be recorded for every mold. These water diagrams may be sketches of the layout or may even be photos of the water layout, but a consistent technique must be established and followed for every mold set. Also on the water setup sheet the in and out temperature of the water should be documented, as well as flow rates and steel temperatures. With all of the above data, troubleshooting a cooling issue versus the baseline process becomes much easier.

33.3.2.4 Mold: Venting

A lack of venting will lead to areas in the mold that may not fill as easily as others. Because these areas do not fill as well the plastic may not be in contact with the mold surface as well as other locations in the mold. Also restricted venting can influence the ability of the part to be packed out against the mold surface.

Verify that the mold is clean and the vents are open. Additional venting in locations with poor gloss consistency may improve the appearance of the part. Lack of venting can create buildup on the mold surface that impacts the gloss levels of the part.

See Chapter 7 for more details on venting.

33.3.3 Gloss Variation Troubleshooting Machine Issues

Gloss issues relating to the machine are usually an issue with machine performance (see Chapter 8).

33.3.4 Gloss Variation Troubleshooting Material Issues

Some specific material-related issues that will lead to gloss are:

- Material type
- Additives

33.3.4.1 Material: Type

Generally speaking, there are materials that will provide a glossier surface than others. In fact, in automotive molding there has been a trend for many components to be molded with as low a gloss level as possible. An example might be a low-gloss versus a high-gloss ABS, each of which is formulated to provide a specific appearance. Make sure that the appropriate material selection accounts for the desired gloss levels.

Often times rubber-modified materials such as ABS, TPO, or ASA will be duller in visual appearance than something like polycarbonate or acrylic, which both tend to be much glossier. Note that some suppliers will offer certain materials in a high- or low-gloss level for specific applications, so be aware of the particular version of the material being molded, and a quick online check will verify if the material is high or low gloss.

Also beware that trying to mold a low-gloss material in a mold with a highly polished surface may result in a variety of surface defects and inconsistency of appearance. Initial material selection needs to account for the desired gloss level much the same as any other desired characteristic.

33.3.4.2 Material: Additives

Many of the additives used in plastics will impact the ability of the material to be molded with a high-gloss surface finish. Fillers such as talc or other minerals often lead to a duller surface finish and high levels of glass will certainly impact the gloss of the part surface. As mentioned in Section 33.3.4.1, rubber modifiers can lead to a part with a less glossy surface finish.

As mentioned in Section 33.3.2.1, use of wood-filled materials can have a major impact on the gloss level of the molded part and also on the polish of the mold. Be watchful for what additives are being used in the material and expect a change when molding material with additives versus unmodified material. Flame-retardant and toughened materials can be prone to outgassing that leads to buildup on the mold surface. Glass-filled materials may lead to erosion of texture leading to gloss variation over time.

Verify that the correct material and additive package are being used.

Case Study: Material Impact on Gloss

In this case parts were being molded with a black TPO. During the molding run the machine operator noticed that the gloss of the parts seemed to be changing. Examination of parts showed that in fact there was a significant gloss and appearance difference between parts from the beginning of the run and later in the run. Investigation showed that a gaylord of incorrect material was moved to the machine and never verified. When the first gaylord

ran out, the feed was switched over to the second gaylord and the incorrect material was loaded and started running. The other material had a different filler loading and resulted in a gloss change.

Always verify materials when they are brought to the machine and prior to loading into the machine.

34 Jetting

34.1 Description

Jetting is a defect where the plastic entering does not form a consistent flow front but rather blasts into the mold in a stream. The stream of plastic will travel across the cavity until the melt stream hits something, at which point the incoming plastic will fill around the jetting. See Figure 34.1 and Figure 34.2 for examples of jetting. Be aware that jetting can produce surface defects that show as blush or splay.

Also known as: worm tracks

Mistaken identities: splay, blush

Figure 34.1 Example of jetting defect

Figure 34.2 Jetting development

34.2 Jetting Troubleshooting Chart

Table 34.1 shows the jetting troubleshooting chart.

Table 34.1 Jetting Troubleshooting Chart

Molding Process	Mold	Machine	Material
high fill velocity	gate location	machine performance	
	gate size		
	gate land length		

34.3 Jetting Troubleshooting

Jetting is a phenomenon that is linked to how the plastic is flowing through a gate. The plastic is not adhering to the mold walls but rather squirts uncontrollably through the mold. It will snake through the cavity leaving behind a worm-like appearance of gloss variation on the part surface.

34.3.1 Jetting Troubleshooting Molding Process Issues

Jetting is typically more related to the mold than the process; however, fill velocity is a process variable that will impact jetting. Conducting a short shot study is valuable when trying to determine if jetting is occurring; there will be cases where the actual defect shows as splay or blush because of how the jetting filled the mold.

34.3.1.1 Molding Process: High Fill Velocity

High injection velocities during first-stage fill will impact how the plastic enters the mold cavity. If the plastic is injected fast the melt flow front may not adhere to the cavity wall but will instead jet across the cavity.

Reducing fill velocity can improve jetting problems. Even though slowing the fill velocity can improve the jetting appearance on a part remember it will increase cycle time and result in adding cost to the molded part. It is always better to address a mold problem rather than process around the issue. Figure 34.3 shows an example of a normal flow front developing from the gate as depicted in Moldflow. A profiled first-stage fill velocity may allow for a slow initial fill with an increased velocity after the melt front is established; this complicates the process, but if nothing else works it may be the only choice.

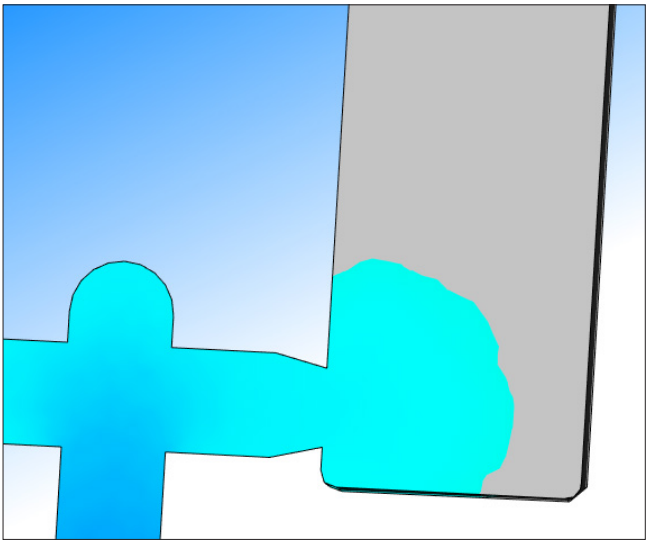


Figure 34.3 Normal flow front developing from the gate (Moldflow)

Flow front development is important as the material enters the mold and continues to fill. Short shots can help determine if the melt front is developing properly; look for inconsistency and abnormal flow patterns. With the way jetting folds the melt

stream back onto itself, air can be trapped leading to splay on the surface in addition to the jetting defect.

34.3.2 Jetting Troubleshooting Mold Issues

Most jetting problems have a root cause in mold design. Important mold-related items are:

- Gate location
- Gate size
- Gate land length

34.3.2.1 Mold: Gate Location

One of the key items of importance concerning jetting is the location of the gate. A key consideration of gate location is not to locate the gate in the middle of the part wall stock. If the gate is located in the middle of the wall stock the plastic entering the cavity will not be able to adhere to the walls of the cavity and may jet as a result. Locate the gates in line with either cavity or core wall steel.

It is desirable to locate a gate so that the melt entering the cavity will impinge on something such as a mold detail or wall section. The term impinging means that the melt stream hits something as it enters the cavity. If there is nothing for the melt to impinge on when entering the cavity one trick is to add an ejector pin, core pin, or other standing steel that protrudes into the part wall stock. This detail will give the melt something to impinge on and breaks up the jetting. This detail can usually be a minimum of 50% of the nominal wall stock for the depth into the wall stock. Be careful with this, though, because the change in wall stock may create a problem with read-through.

34.3.2.2 Mold: Gate Size

Gate thickness can lead to jetting. Frequently, a thin gate leads to jetting; however, there are cases where a thin and wide gate can resolve the problem, so it is important to keep an open mind. Figure 34.4 shows an example of a very thin gate with a long land length.

If jetting is a problem and the gate is excessively thin try to open the gate depth gradually to determine the impact on the jetting. As the gate is thickened the gate seal time increases, which adds cycle time and as a direct result adds cost to the molded part. The case study below shows that troubleshooters must keep an open mind to gate dimensions; do not assume that thicker gates will always improve jetting.

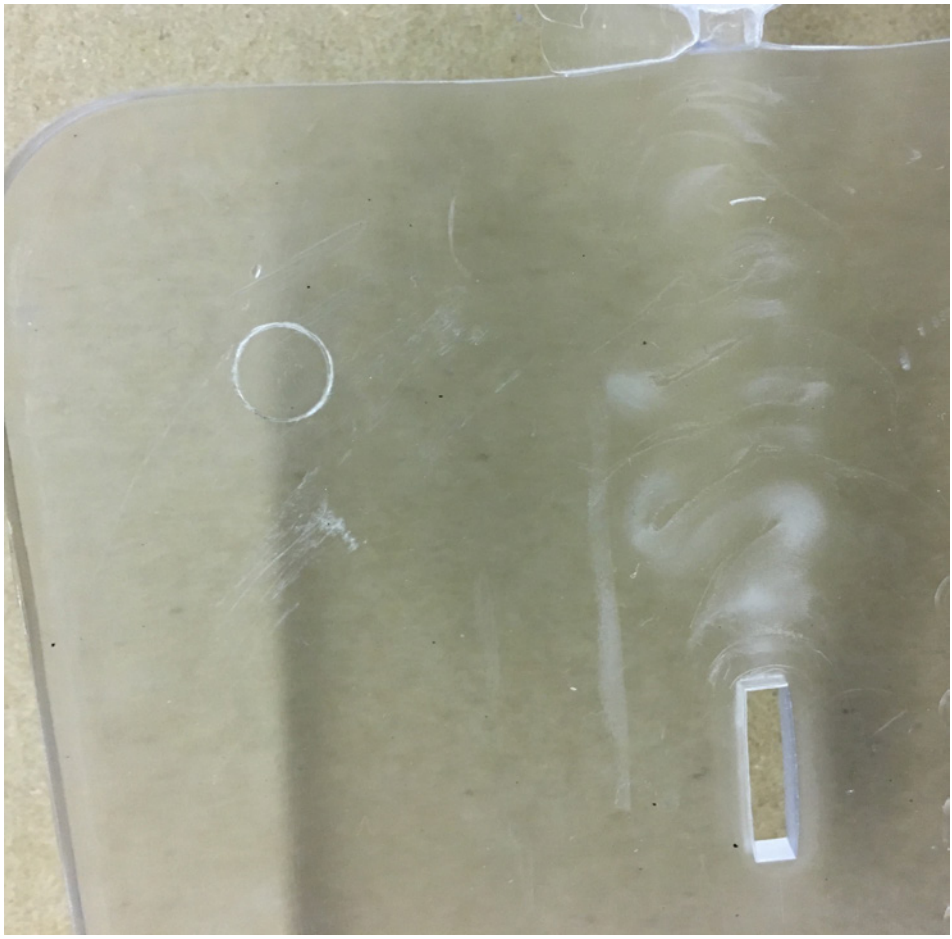


Figure 34.4 Jetting on a polycarbonate part: notice the milky appearance around the jetting

Case Study: Gate Thinning

This case study is one that is eye opening. Normally, thinner gates are more prone to jet but in this case the opposite was true! This polypropylene part originally had two round 0.110-in cashew gates and was experiencing both jetting and pull sinks at the gates. To try to make a change and determine impact the gates were modified to 0.050 × 0.110 in. This change to a thinner rectangular gate eliminated the jetting problem but the pull sinks were still occurring. The gates were again modified to 0.025 × 0.110 in, which eliminated all defects. The change of thickness from 0.050 in down to 0.025 in thick led to an increase in fill pressure from 10,000 ppsi (plastic pressure) to 13,000 ppsi. The thinner gate led to a much-improved process in this case, which again proved that keeping an open mind is critical when troubleshooting.

34.3.2.3 Mold: Gate Land Length

The land length of the gate is the distance that the actual gate thickness is machined into the steel. This distance is from the cavity to where the runner or gate geometry chokes down to its final thickness.

Excessive land length on a gate can impact the likelihood of jetting occurring. The longer the land becomes the more the material is restricted as it enters the cavity and builds up pressure behind the material resulting in a burst into the cavity. Typically land lengths should be kept to a maximum of 0.030 in.

If the land length of the gate is excessive, have a qualified mold maker reduce the land length. This may improve the jetting as well as other issues due to the restrictive land length.

34.3.3 Jetting Troubleshooting Machine Issues

Most causes of jetting are not machine related; however, if the machine is not functioning correctly jetting could be the outcome.

See Chapter 8 on machine performance.

34.3.4 Jetting Troubleshooting Material Issues

Given the wrong gate and molding process almost all materials can be made to jet.

35 Part Sticking on Cover

35.1 Description

Parts can tend to stick on either the cover or the ejector side of the mold. Solutions can be different depending on which side of the mold the part is sticking on.

35.2 Part Sticking on Cover Troubleshooting Chart

Table 35.1 shows the part sticking on cover troubleshooting chart.

Table 35.1 Part Sticking on Cover Troubleshooting Chart

Molding Process	Mold	Machine	Material
high second-stage pressure	mold damage	process control	moisture content
melt temperature	lack of draft		material type
high mold temperature	surface finish		lack of release agent
mold open breakaway speed	vacuum		
	cavity balance		
	buildup		
	ejector half detail		

35.3 Part Sticking on Cover Troubleshooting

Sometimes a part will completely stick to the cover side of the mold and other times it may stick and distort but stay on the ejector half of the mold. Often, sticking to the cover will cause cracking noises during mold open. If cover sticking is suspected stop the mold before ejection and look for areas of the part that have pulled away from the ejector because this indicates that the part was sticking in the cover and tried to pull away from the ejector.

35.3.1 Part Sticking on Cover Troubleshooting Molding Process Issues

Some process conditions that can lead to parts sticking to the cover are:

- High second-stage pressure
- High melt temperature
- Low or high mold temperature
- Mold open breakaway speed

35.3.1.1 Molding Process: High Second-Stage Pressure

Sticking on the cover side of the mold often indicates high cavity pressure. If the cavity pressure is high the part may not be shrinking enough to be effectively pulled away from the cavity. High second-stage pressure can over-pack plastic into details in the mold and molds with a lot of cover side detail will be more prone to over-packing.

Verify the second-stage pressure against the documented process and make adjustments based on it. When evaluating pressure, it should always be viewed as plastic pressure, which means that the intensification ratio of the machine must be accounted for.

35.3.1.2 Molding Process: High Melt Temperature

If the actual melt temperature is running on the high side it can impact sticking for two reasons:

1. High melt temperatures can lead to increased material degradation, which can make a material more prone to stick.
2. High melt temperatures lead to an enhanced ability to distribute cavity pressure in the mold. See Section 35.3.1.1 for more on high pressure concerns for sticking.

Confirm that the melt temperature meets the recommended settings of the material supplier. If the melt temperature is running high verify that settings that impact melt temperature are correct including:

- Barrel temperatures
- Back pressure
- Screw recovery speed

35.3.1.3 Molding Process: Mold Temperature

The actual steel temperature of the mold can lead to sticking concerns. If the mold temperature is too high the material may not shrink enough to effectively release from cover details. High mold temperatures also lead to less pressure drop across the cavity and allow the part to be packed out more effectively potentially leading to sticking to the cover. Normally if mold temperatures are too cold sticking problems will be more likely to occur on the ejector side of the mold.

There may be cases where the cooling capabilities of the mold must be improved to prevent sticking (see Chapter 14 on mold cooling).

35.3.1.4 Molding Process: Mold Open Breakaway Speed

If the initial mold open speed is excessively fast sometimes the part will have more tendency to stick to the cover side of the mold. There are cases where there is a vacuum between the part and the mold and a slower speed will allow the vacuum to break. Also fast breakaway speeds can cause a part to become angled onto mold details and lead to sticking.

If the part is sticking to the cover try a reduced breakaway speed to determine if the sticking improves. If it does improve the problem, utilize the STOP process to work through any root causes that may be resulting from the mold itself such as undercuts. Slowing the breakaway speed may end up as processing around another root cause.

35.3.2 Part Sticking on Cover Troubleshooting Mold Issues

There are many potential mold-related causes of part sticking including:

- Mold damage
- Lack of draft
- Surface finish
- Vacuum
- Cavity balance
- Buildup
- Ejector half detail

35.3.2.1 Mold: Damage

Any damage to the parting line or the cavity of a mold can lead to sticking problems and any damage that results in an undercut in the mold will restrict the part from releasing away from the mold.

Examine the cavity for any dings, scratches, or gouges that need to be repaired. Also feel along the parting line for burrs that are creating an undercut. If damage to the mold is discovered it must be repaired to resolve the sticking problem.

Avoiding mold damage is the key reason why steel tools should not be used inside the cavity. The risk of damaging the mold surface with the steel is to great so part removal should normally use brass tools. Keep in mind that even soft brass tools can damage a high polish or textured surface. Everyone must be careful when using any tool to remove parts from a mold.

35.3.2.2 Mold: Lack of Draft

Molded plastic parts need draft to allow the part to be removed from the wall. Draft is the term used to describe the angle of any walls, cores, or bosses that are in the line of draw for the mold. Because draft angles open up as the part detail approaches the nominal wall stock the part will quickly be separated from contact with the mold. If the detail was designed at true zero draft the plastic would remain in contact with the mold for the total length of the detail. Without draft the only thing that allows the part to be freed from the mold surface is shrink.

Depending on the material different draft angles will be required. Stickier materials need additional draft and locations where the part is shrinking onto the steel of the mold require more draft than shrink-away areas.

Caution with draft must be applied because the angle will impact the thickness of the detail. A deep rib will release best with extra draft but the wall stock of the rib may become too thin to effectively fill. Draft analysis should be conducted on all part designs to verify the correct draft angle is being applied, and evaluate areas where draft may cause wall stock problems.

35.3.2.3 Mold: Surface Finish

As texture depth increases, the amount of draft required to remove the part also increases. Because texture is a series of undercuts, the plastic is stuck and must have sufficient draft to fall away from the texture. Lack of draft on texture will lead to either a sticking part or scuffing of the texture.

If a mold is to be textured the depth of the texture must be known to allow accommodation of draft angles. Texture companies can help establish the required draft for a given texture style. A general rule of thumb is to add 1° of draft for every 0.001 in of texture depth. If the appropriate draft is not added to the mold design

the depth of the texture may need to be reduced before parts can be successfully molded without sticking and scuffing.

When it comes to polished surfaces a 320 draw polish can provide a good polished surface to provide adequate release. There are many cases, especially with softer materials such as polypropylenes, TPE, or TPU, where an increase in polish level will lead to more of a vacuum effect between the plastic and the mold surface, leading to increased sticking.

35.3.2.4 Mold: Vacuum

On highly polished molds there will be cases where the plastic has an adhesive force on the steel creating a vacuum against the steel. This is most typical with soft materials and makes sticking a potentially major concern.

If the part is sticking due to high-polish vacuum issues, the first option to evaluate is the impact of a less polished surface. There are cases when the part is brought up to a polish level that is greater than what is required, so if the polish can be reduced the part may come free. A second option would be incorporating an air poppet in a hidden location. The air poppet will allow air to flow between the part and the mold surface breaking the vacuum.

Understand the polish requirements for a given part and determine if that will work for the chosen material. If molding a material such as TPU a polished surface will create major sticking problems, but a sand-blasted finish will allow easy removal of the molded part.

35.3.2.5 Mold: Cavity Balance

A multicavity mold should be balanced within 3% for all cavities. If the mold is imbalanced between cavities the process that each cavity experiences will be different. Unbalanced molds will have different cavity pressures, which in turn can lead to different shrink rates and make some cavities stick while others do not.

Run a cavity balance study to determine what the balance of the mold is. It should be less than 3%. If the mold is showing greater than 3% imbalance, investigation into the root cause must be undertaken.

For more details see Chapter 12 on cavity balance.

35.3.2.6 Mold: Buildup

Sometimes buildup from material by products will cause a part to stick. The buildup may be sticky to the material or actually even create slight undercuts if thick enough.

Verify that the mold surface is thoroughly cleaned. If buildup is an ongoing issue the mold should be vented better. Utilize the proper cleaner combination to remove

the buildup. Some material buildup can be cleaned off better with resin remover, citrus-based mold cleaners, or a mold cleaner such as Zapox. There are also cases with materials such as TPU where the mold may actually need to be bead blasted to remove the buildup sticking issues.

Case Study: TPU Sticking

In this case a part was being molded from an 85-shore durometer TPU. This TPU tended to become very sticky when degraded from either moisture or residence time. The worst aspect of this problem was that when anyone molded a part with degraded material a buildup was left behind that would not clean off with any commercial cleaners. To remove the buildup and keep parts from sticking, the mold had to be removed from the machine and bead blasted. After blasting, the surface would be fine until someone shot degraded TPU again.

The solution to this problem was training of all process technicians to observe the viscosity of the material when purging. If the material was runny it was likely degraded and they needed to purge longer.

35.3.2.7 Mold: Ejector Half Detail

If the ejector side of the mold does not have adequate detail to retain the part during mold opening the part will tend to stick to the cover. Proper selection of the "A" versus "B" side is important during design and the ejector side should have detail that the plastic is shrinking onto.

If the ejector side does not have adequate detail there are some tricks that can improve the sticking problem including:

1. Undercuts ground into the ejector side.
2. Texture added to the ejector side.
3. Sand-blasted finish on the ejector side.
4. Preloaded push pins on the cover half to push the part off.

35.3.3 Part Sticking on Cover Troubleshooting Machine Issues

Process control is the main machine concern relating to sticking in the mold. All process settings on the machine must be accurately controlled by the machine. If the machine is not controlling the set points it is the same as entering the wrong set points. The machine must provide the temperatures, pressures, and speeds accurately.

Check that actual values on the machine correspond with the set points. Things to watch for include temperature zones running above set point and pressures that are not quickly reaching set point after velocity to pressure (V to P) transfer.

There are cases where the set pressure does not match the gauge pressure or the pressure reading on the machine controller. Which one to trust in this case? An auxiliary gauge or a pressure transducer connected to a process monitoring system will provide an accurate measure of actual pressure. It may be necessary to replace the gauge and the pressure transducer of the machine to provide accurate results.

35.3.4 Part Sticking on Cover Troubleshooting Material Issues

Some of the potential material factors that influence sticking include:

- Moisture content
- Material type
- Lack of release agent

35.3.4.1 Material: Moisture Content

Some materials when processed with moisture content will degrade and become sticky. Whenever processing a hygroscopic material, it is critical to ensure that the material has been dried to the required moisture percentage.

Verify moisture content with a moisture analyzer. If moisture content is high allow additional drying time to completely dry the material.

There are cases where sticking is occurring and the machine is shut down to polish or work on the mold. During this downtime the material continues to dry and upon restarting the machine the sticking problem is resolved. Unfortunately, the mold is often blamed as the root cause but in many cases the extra drying time is what corrected the problem. When dealing with sticking problems verify the moisture content of the material before working on the mold.

For details on drying refer to Chapter 9.

35.3.4.2 Material: Material Type

Some materials are prone to sticking under varying conditions. Materials such as TPU, polyesters, and polycarbonates are examples of materials that have an increased tendency to stick.

As mentioned in Section 35.3.4.1 moisture can be a big issue for sticking. A reaction called hydrolysis can occur when processing with excess moisture content; hydrolysis is a breakdown in the molecular weight of the material. Materials that

are prone to suffering from hydrolysis such as those listed above need to be dried thoroughly and kept dry to avoid sticking problems.

Semi-crystalline materials will experience more shrink than amorphous materials, which will allow the plastic to shrink away from cover side detail (providing the shrink on core conditions were designed into the ejector side of the mold). This increased amount of shrink can help or hurt sticking conditions and usually interacts with the cooling rate that the part is experiencing while in the mold.

Also be aware that different materials will react to different mold surfaces and may stick worse because of the mold surface finish. Most times soft materials such as TPEs or TPUs will tend to stick to polished surfaces due to a vacuum effect between the soft material and the mold surface. Soft materials will often release much better with a bead-blasted surface. If a polished surface is required an air poppet in a hidden location may help break the surface vacuum.

35.3.4.3 Material: Lack of Release Agent

If the material of choice does not have enough release agent in its additive package, parts will be more likely to stick. If sticking coincides with a lot change it may indicate a potential material problem.

There will be cases where a material supplier offers an enhanced release package on the same material. The correct material choice must be identified up front. If severe sticking is occurring the material should be reviewed with the customer for potential replacement options.

There are cases where a secondary additive such as zinc stearate is added to a material prior to molding. If the part being molded uses a release agent additive verify that it is being used and at the correct loading.

Remember that manually spraying mold release on a mold to eliminate sticking issues is not solving the problem. Giving the mold a quick spray at startup is common but if the mold needs to be sprayed on a regular basis to avoid sticking the cause of the sticking should be identified and corrected. As soon as a person forgets to spray the mold after x number of shots a part is going to stick and cause downtime and potentially damage the mold. Whenever using a spray mold release for startup sticking make sure to verify that it is the correct type, i. e. paintable or food grade where required.

36 Part Sticking on Ejector

36.1 Description

Parts can tend to stick on either the cover or the ejector side of the mold. Solutions can be different depending on which side of the mold the part is sticking on. Figure 36.1 shows plastic stuck on core detail in a mold.

Figure 36.1 Plastic stuck on core of mold

36.2 Part Sticking on Ejector Troubleshooting Chart

Table 36.1 shows the part sticking on ejector troubleshooting chart.

Table 36.1 Part Sticking on Ejector Troubleshooting Chart

Molding Process	Mold	Machine	Material
low or high second-stage pressure	surface finish	machine performance	material type
low or high mold temperature	lack of ejection	ejector plate	moisture content
long or short cooling time	lack of draft	robot handling	release agents
fast ejection speed	buildup		
low or high melt temperature	slides or lifters		
	damage, erosion, or burrs		
	lack of cooling		

36.3 Part Sticking on Ejector Troubleshooting

There are many causes of part sticking on the ejector half of the mold and unfortunately many settings can cause problems if set too low or too high. If relying on processing alone to resolve a sticking problem the result will probably be a tight process window, so dig deeper and find ways to improve the mold for best results.

36.3.1 Part Sticking on Ejector Molding Process Issues

Possible molding process related sticking causes are:

- Low or high second-stage pressure
- Low or high mold temperature
- Long or short cycle time
- Fast ejection
- Low or high melt temperature

36.3.1.1 Molding Process: Second-Stage Pressure

Second-stage pressure can create sticking problems on the ejector side of the mold if it is too low or too high.

When second-stage pressure is set too low, the plastic may shrink excessively onto the ejector half of the mold. This excessive shrink can result in a part that is stuck too hard to eject. Try increasing second-stage pressure to better pack out the part and reduce shrinkage. Evaluate if the increased pressure improves the sticking. If excessive shrink is the problem, fill only shots may also tend to shrink to the ejector side of the mold.

If second-stage pressure is too high and there is a lot of recessed part detail on the ejector half of the mold the plastic may be over-packed into the detail. This over-pack situation can lead to reduced shrink of the plastic and may cause the part to stick. If recessed details seem to be sticking (look for distortion) try reducing the second-stage pressure to determine if the sticking improves.

When troubleshooting verify that second-stage pressure is set to the documented process. Also ensure that the machine is actually achieving set pressure. Cavity pressure transducers will help to ensure that the actual results in the mold are accurate to the established process.

36.3.1.2 Molding Process: Mold Temperature

When it comes to sticking on the ejector half of the mold, the steel temperature can be the root cause. This is another factor that can either be too low or too high depending on the part and process.

When mold temperatures are too low the part may not be packed out as well, which can lead to additional shrink resulting in sticking. Increased shrink can also occur with high mold temperature and semi-crystalline materials. With semi-crystalline materials larger crystals will form, which can lead to increased shrink. This impact of either low or high can make mold sticking a difficult-to-solve problem.

If the mold temperature is too high the part may be over-packed leading to sticking in part recesses such as ribs or bosses. Warmer molds lead to less pressure drop across the cavity and may result in higher cavity pressures near the end of the cavity causing sticking.

Verify the thermolator set temperature, actual mold steel temperature, and the part ejection temperatures against the documented process. If the thermolator is set incorrectly make the appropriate adjustment. If the thermolator is set correctly but the steel or part ejection temperature is wrong then a deeper dive into the cooling parameters is needed (see Chapter 14 on mold cooling).

36.3.1.3 Molding Process: Long or Short Cooling Time

On some occasions, running with too much cooling time will allow the molded part to shrink too much to the ejector detail resulting in a stuck part. Processes sometimes are run with more cooling time than is required to produce a quality part. Generically once the part is below the materials glass transition temperature (many molders may also use the heat deflection temperature or HDT) it can be ejected without warp; note that using a single data point for choosing part ejection temperature may be challenging. Many molders end up using excess cooling time for no good reason. Let the part data make the decision, and if the part is stable at the faster cycle time then the process can run here.

If the cooling time is too short the plastic may be too soft to eject from the mold. Make sure that the part has adequate cooling time to prevent sticking. In an exaggerated condition the ejector pins will push through the soft part leaving the part stuck on the ejector.

Evaluate the running process against the documented process, and correct if cooling time is set incorrectly. During process development look for opportunities to run the mold faster and potentially eliminate excess shrink on the mold.

36.3.1.4 Molding Process: Fast Ejection

Fast ejection speeds can lead to issues with a part binding on the ejector side of the mold. There are cases when the fast ejection speed can cause the part to cock on the mold resulting in sticking. Also fast ejection speeds may cause a part to adhere to a lifter and not travel evenly away from the ejector half of the mold. Also remember that there are cases where a vacuum forms between the mold and the part; slower ejection speeds may allow this vacuum to break and allow better release of the part.

If slowing the ejection speed improves the release of the part from the ejector side of the mold use the STOP process to determine if any other root causes are leading to the sticking problem. Do not slow the cycle time to process around a mold issue! Adding undercut keepers to limit the opportunity for the part to travel with a lifter may lead to complete resolution of the problem without slowing down the ejectors.

36.3.1.5 Molding Process: Melt Temperature

With a part sticking on the ejector side, either low or high melt temperature can lead to problems. A low melt temperature may result in a lack of cavity pressure, which in turn can lead to extra shrink on the ejector. High melt temperatures can lead to either more or less shrink depending on the mold; in some cases higher melt temperature leads to less pressure and less shrink and in other cases it leads to increased shrink due to higher crystallinity from slower cooling.

If other factors do not seem to influence the sticking problem try adjusting melt temperature to determine impact. If changing melt temperature make sure to re-check gate seal and final part dimensions.

36.3.2 Part Sticking on Ejector Troubleshooting Mold Issues

There are many mold-related issues that can cause part sticking on ejector including:

- Surface finish
- Lack of ejection
- Lack of draft
- Buildup
- Slides or lifters
- Damage, erosion, or burrs
- Lack of cooling

36.3.2.1 Mold: Surface Finish

Finish on the ejector half of the mold is often overlooked and as a result sticking can be a major problem. In many cases molds will have EDM or machining marks left on the ejector half of the mold that prohibit ejection. Normally a 320 draw polish will provide good release properties from the mold. In many cases too much polish can lead to sticking issues, but a good draw polish is often the key to providing release. Be sure to examine molded parts with a flashlight to look for EDM or machining marks on all details.

36.3.2.2 Mold: Lack of Ejection

If a mold is designed without adequate ejection the part will stick to the ejector side of the mold. Ejection must be incorporated during mold design and must be balanced with cooling requirements as well as designed support pillars.

Where possible it is often advantageous to increase the size of the ejector pin to provide more ejection surface area as well as avoid pin deflection from thinner pins. Use of straight lifters can allow for increased surface area for the ejection force to be distributed against. Ensuring adequate ejection will help eliminate sticking, pin push, and even improve cycle times.

36.3.2.3 Mold: Lack of Draft

Every material has specific requirements for draft angles to provide release from a mold. Draft needs to be incorporated into the design of the part and the mold to

account for sticking. Draft angles allow a molded part to "break away" as it is ejected and allow clearance as the part moves away from the core details of the mold. Review standard requirements for the material being molded and apply the appropriate draft to the part.

36.3.2.4 Mold: Buildup

During molding, gases from the material will be formed as the plastic is melted, and these gases can result in a buildup or plate out on the mold surface that can lead to sticking. The mold surface should always be kept clean and if buildup is an ongoing problem, venting improvements should be made (see Chapter 7 on venting and Chapter 19 on buildup).

36.3.2.5 Mold: Slides or Lifters

If a lifter or slide does not travel enough the undercut detail may not exit the component. All lifters and slides should be reviewed during mold design to determine if adequate travel is achieved. During mold setup it is important to verify that the proper amount of ejection stroke is being used to allow lifters to reach a full forward position.

It is important for mold components to pull away from the part allowing clearance between the component and the part. There are cases where even though the component travels far enough the part will travel with the component, i. e. it sticks to a lifter rather than the core steel. In these cases it may be necessary to add a rib or other undercut to the core steel to prevent the part from moving with the lifter. Whenever possible, lifters should be designed to travel in opposing directions, which will help balance the tendency of the part to travel with lifters in one direction.

36.3.2.6 Mold: Damage, Erosion, or Burrs

Any time the mold is damaged, sticking could be the result. The damage to the mold may be a matter of a burr on the mold that is creating an undercut that traps the part; any burrs should be removed to prevent sticking. Erosion can occur in areas of the mold that have inadequate venting, where the trapped gases will start to erode the steel over time, which can lead to a surface finish that is rough and will trap the molded part.

Other items of damage may include hydraulic cylinders on either core pulls or hydraulic ejection. Verify that any cylinders used for part ejection are not leaking and are connected properly.

36.3.2.7 Mold: Lack of Cooling

A mold that is lacking in cooling will often lead to problems with ejection. If the part cannot be adequately cooled it will be difficult to eject from the mold. Typically, the unfortunate result of lack of cooling is an extended cycle time that allows ejection of the part without damage or sticking problems.

Cooling must be addressed during the design phase of the mold. Careful evaluation of water line coverage is important and must be balanced with providing ejection. It is also important to understand that if the cooling system of the mold is too large for the available mold temperature control unit it will be impossible to achieve turbulent flow and cooling will be compromised (this further highlights the importance of the 4M thought process where each of the key items influences the others).

36.3.3 Part Sticking on Ejector Troubleshooting Machine Issues

Some potential root causes for sticking include:

- Machine performance
- Ejector plate
- Robot handling

36.3.3.1 Machine: Machine Performance

See Chapter 8 on machine performance.

36.3.3.2 Machine: Ejector Plate

If the ejector plate on the machine is not moving in an even fashion it can lead to the ejector plate of the mold being cocked. If the ejector plate is cocked the ejection force will not be evenly distributed against the part, which can cause the part to lock onto the ejector.

Verify that that the knock-out bars are uniform in length. Bars that are different lengths will lead to a cocked ejector plate. Check the ejector plate on the machine and check that the bushings are not worn. A wobbling ejector plate is a strong sign that something is wrong.

Verify that the ejector bars are the proper length. If bars are too short the lifters may not reach full travel resulting in an undercut that traps the part on the mold. Proper bar length should always be used when the mold is set up to maximize the ejection stroke available.

Another factor to check is that the correct number of knock-out bars are being used. One center knock-out bar on a 55-ton press may work fine but trying to push

an ejector plate on a 300-ton with one knock-out bar will likely lead to plate-cocking problems.

36.3.3.3 Machine: Robot Handling

When a robot removes a part from a mold there can be problems getting the part free from the mold. Two potential causes of sticking that relate to the robot are:

1. The robot pushes the part back onto lifters and locks the part onto the lifters.
2. The robot does not pull straight back away from the mold resulting in part detail getting trapped on the mold.

Carefully watch the robot as it makes its movements while extracting a part. Unusual movements can highlight a step in the robot program that can be optimized. Adding additional grippers, gripper fingers, or standoffs to the end of arm tooling may stabilize the part and allow for improved handling.

36.3.4 Part Sticking on Ejector Troubleshooting Material Issues

Potential material causes of sticking include:

- Material type
- Moisture content
- Release agents
- Viscosity

36.3.4.1 Material: Material Type

Some materials will be more prone to sticking in the mold including:

- TPU
- Polyesters
- Polycarbonate

It is critical for all materials to build the mold with the appropriate draft angles for the intended material. Check the recommended draft of the material supplier for both shrink on and shrink away conditions.

With TPU and polyesters any process-related condition that degrades the material will magnify the sticking concern. Many times with these types of materials the process can be running and all of a sudden the ejector pins punch through the part indicating that the material has gotten wet! With both TPU and polyesters make sure that the machine is thoroughly purged with dry material before injecting the material into the mold.

Case Study: Sticking TPU

TPU is thermoplastic polyurethane, which becomes very sticky when allowed to degrade. TPU is extremely sensitive to moisture content, temperature, and residence time.

In this case the part was having off-and-on sticking problems. When sticking would occur it would be so bad that the material would leave behind a residue on the mold surface. To effectively eliminate the residue the mold would need to be sand blasted.

Evaluation of startup procedures showed that people were having two different issues:

1. Starting the press with wet material

   This guaranteed that the part would stick every time. All technicians were trained to verify the moisture content before starting the mold.

2. Not purging the barrel thoroughly prior to start up

   Because the material was so sensitive to temperature and residence time the technicians had to learn to appreciate how little down time they could afford with this particular mold.

Clear training and startup procedures were critical to avoiding an ongoing problem with this mold.

36.3.4.2 Material: Moisture Content

Most of the materials mentioned in Section 36.3.4.1 will be sensitive to moisture content. Ensure that materials are adequately dried before molding (see Chapter 9 on material drying for more details).

36.3.4.3 Material: Release Agents

Most molding materials have release agents and lubricants added to help with processing. If a given lot of material starts to stick it may indicate that the level of release agent is not the same as in previous lots. If adding anything to a material that may contain additional release agents or lubricants verify that the correct let-down ratio is being used for the additive.

Use of a spray release may be required during process startup but is poor practice for ongoing production. If a mold will not run without using spray mold release on a continuous basis there is a problem that must be identified and corrected. Adding an operator with a can of mold release to get parts to come off a mold is an expensive Band-Aid to get a mold to run; instead, find the root cause of the sticking and correct it.

36.3.4.4 Material: Viscosity

If the viscosity of the material changes for any reason it can impact how the plastic is packed and cooled in the mold. Lower-viscosity (easier flowing) material will allow for better transfer of pressure throughout the cavity, which can impact shrink and cooling. Bear in mind that the incoming viscosity of the material may change but changes in the process will also influence viscosity. Always verify the following:

- Melt temperature
- Barrel temperatures
- Back pressure
- Screw recovery rate
- Barrel size
- Intensification ratio
- Fill rate

Any of the above process conditions may lead to a viscosity change that could impact how the part sticks in the mold.

37 Pin Push

37.1 Description

Pin push is a defect where the ejector pins show glossy or white imprints on the class A surface of the part. See Figure 37.1 for an example of pin push.

Also known as: pin marks

Mistaken identities: read-through, gloss variation

Figure 37.1 Pin push

37.2 Pin Push Troubleshooting Chart

Table 37.1 shows the pin push troubleshooting chart.

Table 37.1 Pin Push Troubleshooting Chart

Molding Process	Mold	Machine	Material
high or low second-stage pressure	sticking	machine performance	release agent
low or high mold temperature	lack of ejection	ejector plate	moisture content
high melt temperature	lack of draft	velocity to pressure transition	fillers
fast ejector speed	cavity balance		
cooling time			

37.3 Pin Push Troubleshooting

Pin push usually is a defect where the ejection force on a part distorts the wall of the part. Many factors can impact the tendency of the wall stock to be distorted resulting in pin push.

37.3.1 Pin Push Troubleshooting Mold Process Issues

Molding concerns for pin push include the following:

- High or low second-stage pressure
- Low or high mold temperature
- High melt temperature
- Fast ejector speed
- Cooling time

37.3.1.1 Molding Process: High or Low Second-Stage Pressure

Second-stage pressure has a large impact on how a part sticks or shrinks onto a mold. Depending on the detail on the ejector side of the mold both over-packing and under-packing of the plastic can lead to pin push.

Over-packing of the plastic can reduce the amount of shrink on the molded part, which can in turn lead to ribs and other details that are packed so tight that they

stick. The sticking in turn can lead to visual problems with pin push as the ejector pins force the part from the core steel.

Under-packing can lead to excess shrink that in turn can cause the molded part to stick to details on the core side of the mold. This excess shrink can result in pin push because the part has shrunk and stuck on detail.

When working on pin push try both raising and lowering second-stage pressure. Determine if there is any improvement in the pin push on changing the pressure. Remember in troubleshooting that a change that makes something worse can tell a great deal about the impact of the setting.

37.3.1.2 Molding Process: Low or High Mold Temperature

As with second-stage pressure, mold temperature can impact pin push if running too low or too high. Usually, a too-high mold temperature will lead to more pin push because the ejector pins are impacting a softer, more pliant surface. Examine the part out temperature with a surface probe or a thermal imaging camera. Beware of hot spots that may cause pin push from being too pliant during ejection.

If a mold is running too cold the part may shrink too much prior to ejection. Even though the wall stock is more solidified the extra shrink on the core may result in pin push.

37.3.1.3 Molding Process: High Melt Temperature

When running with a high melt temperature the molded part will be softer and may be more likely to display pin push. Injection molding is very dependent on thermal management and running with excess melt temperature will cause problems with managing the heat. If the molded part is too hot the odds of seeing some surface distortion or stress from the ejector pin will be magnified.

Verify that the melt temperature is running in line with the documented process. Also compare the melt temperature to the recommended setting of the supplier. If the melt temperature is high investigate the potential root causes. There are cases where melt temperature was raised to accommodate other problems such as thin wall areas that are difficult to fill. If possible, resolve the root cause of the high melt temperature and see if that improves the pin push.

37.3.1.4 Molding Process: Fast Ejector Speed

If the ejectors are travelling too fast, pin push may be the result. Some parts do not do well with trying to eject as fast as the machine allows. Look at slowing down the ejector speed to determine if the pin push is improved. Slowing down the ejectors may allow the part to ease off the mold rather than using just brute force. A slight increase in cycle time will result from slowing down the ejectors but if it resolves the defect it may be necessary.

37.3.1.5 Molding Process: Cooling Time

The length of time that a part spends in the mold cooling will potentially impact pin push in two different ways:

1. Excess cooling time may lead to increased shrink and the part may stick to the ejector half of the mold leading to pin push during ejection.
2. If cooling time is too short the parts will be ejected in a state that cannot resist deformation from the force of ejection, which can lead to pin push.

Check the part ejection temperature and if it is running higher than the heat deflection temperature (HDT) of the material the cooling should be increased to allow more heat energy to be removed from the parts prior to ejection. If the parts are ejecting at a cool temperature try to reduce cooling time to determine if pin push is improved. Whenever cooling or overall cycle time is changed be sure to verify dimensions and any other key parameters.

37.3.2 Pin Push Troubleshooting Mold Issues

Some potential mold-related concerns are:

- Sticking
- Lack of ejection
- Lack of draft
- Cavity balance

37.3.2.1 Mold: Sticking

If the molded part sticks on the ejector half of the mold, pin push may be a resulting defect. Because the part is sticking, the force that is applied from the ejector pins can cause pin push defects.

Some of the key things to examine for possible sticking causes include:

1. Undercuts on the ejector side of the mold

   These undercuts could be a rolled parting line edge or other damage to the mold steel. Also examine if pullers have been ground in the ejector, which may be too deep, which can lead to sticking. If any of the above is causing sticking, address the issue in the steel.

2. Texture on the ejector side of the mold

   If there is texture on the ejector half of the mold, it must have an appropriate amount of draft to allow the texture to fall away from the mold surface.

3. Machining/cutter marks left on ejector half of mold

There are cases where the machining marks must be polished out of the mold to get adequate release. Many times, a draw polish will help the part release easier and eliminate pin push.

Case Study: Polypropylene Pin Push

In this case a polypropylene part was being molded but the part had pin push. Various process settings were tried to improve the pin push problem without success. A tool maker reviewed the part and suggested that a draw polish may help the problem. He did a paper draw polish on the core of the mold, which solved the problem. Another example of "do not process around a mold problem".

37.3.2.2 Mold: Lack of Ejection

Every molded part will require a certain amount of force to eject from the mold. This force is translated to the part through the ejector pins and lifters. If there is a lack of ejection surface area the force on the pins will be higher and more likely to create pin push.

When designing a mold it is critical to get adequate ejection surface area in the mold. It is best to push a part detail off the core rather trying to pull a part detail from the core. Whenever possible get pins, lifters, or sleeves under a part detail to push with. Use large diameter pins where possible to avoid deflection and spread the ejection force over a greater area. If there is a rib detail that a pin cannot be located under, make sure that there is adequate ejection around the detail. On sub gated or cashew gated parts locate ejector pins inside the part near the gate to provide a good push as the gate is being separated.

Keep in mind that straight lifters can be used for ejection and give much more surface area to push with. Look for areas in a mold where a straight lifter may make sense.

37.3.2.3 Mold: Lack of Draft

Draft is really as much a part design issue as anything but it also must be addressed during mold design. Trying to remove a part from the mold with no draft means that the part will drag against wall stock the whole length of ejection. The force required to eject a part lacking in draft will increase and either result in a stuck part or pin push (or pins pushed right through the part).

Review the recommended draft requirements for the material being molded and stick to the recommendations. There are cases where the customer pushes for less than the appropriate amount of draft and they usually end up causing problems. In some cases multiple attempts at polish end up opening up the draft to something that allows the part to finally release.

37.3.2.4 Mold: Cavity Balance

If a multicavity mold is not balanced between all cavities there will be differences in the molding process between each cavity. Nonuniform filling can lead to over-packing in some cavities and under-packing in others, which can result in pin push in some of the parts whereas others are fine.

Trying to accommodate for fill imbalance with the process will close the process window for the mold severely. It is a much better decision to fix the mold and show every cavity the same processing effects.

See Chapter 12 on cavity balance for more information.

37.3.3 Pin Push Troubleshooting Machine Issues

Machine impact on pin push includes:

- Machine performance
- Ejector plate
- Velocity to pressure (V to P) transition

37.3.3.1 Machine: Machine Performance

The molding machine must be capable of repeatedly achieving the process that has been set. If the machine is varying on any of a number of process parameters such as pressures and ejector settings, pin push can be the outcome.

See Chapter 8 on machine performance for more details.

37.3.3.2 Machine: Ejector Plate

Pin push can result from a nonuniform ejection force being applied to certain pins in the mold. This issue is most likely to stem from using knock-out bars that are not the same length. If the knock bars are uneven the ejector plate will push on the ejector plate in the mold with uneven force and possibly cause pin push. Always ensure that the knock-out bars are of the same length and screwed in equally.

Another potential cause of uneven ejection could be damage to the ejector plate or ejection cylinder itself. If the ejector plate is becoming cocked as it moves forward, the result will be uneven force on the ejector plate of the mold. Watch the ejector plate of the machine for wobble or cocking as it travels forward. Excess wear in the ejection system of the machine can lead to a variety of ejection problems with the mold and part. In some cases the bushings for the ejection guides are worn out and in others the ejector holes in the platen can become elongated if the knock bars are not tied into the ejector plate.

37.3.3.3 Machine: Velocity to Pressure Transition

An often-overlooked factor that can lead to pin push is how well the machine transfers from first-stage velocity control to second-stage pressure control. There are many cases where this transition is not responsive and instead of immediately reaching the desired pressure the machine will overshoot. Figure 37.2 shows a process monitoring screenshot where the machine injection pressure spikes way above set point. The cavity pressure curve shows the resulting spike in actual cavity pressure. This type of poor transfer can lead to over-packed parts that in turn lead to pin push.

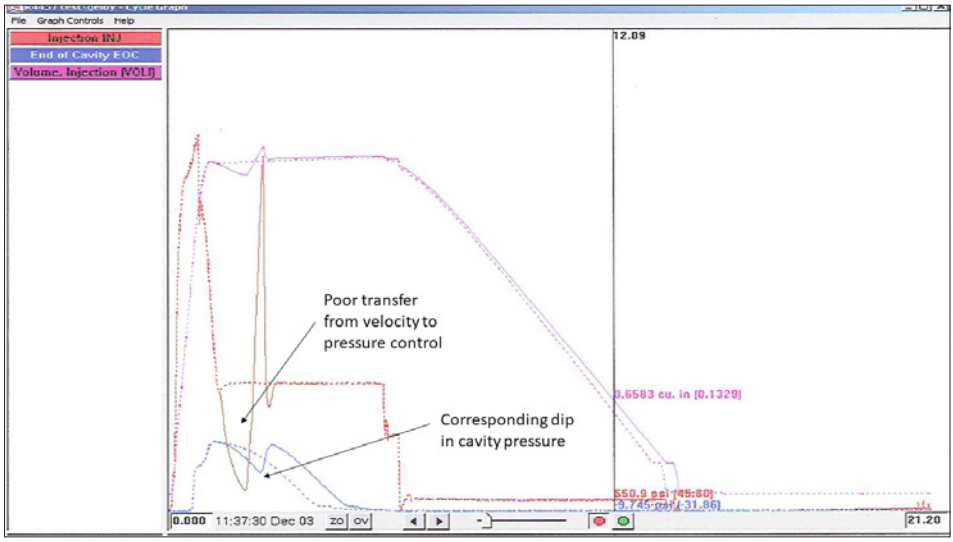


Figure 37.2 Process monitoring showing overshoot at velocity to pressure transfer

37.3.4 Pin Push Troubleshooting Material Issues

Some potential material-related causes of pin push may be:

- Release agent
- Moisture content
- Fillers

37.3.4.1 Material: Release Agent

The majority of molding materials have some form of lubricants and/or release agents to improve their moldability. There are many cases where the same material comes in an enhanced release formulation that may work better for a given

application. During development it is worth evaluating if the enhanced release material is allowed for the application and determining if it processes better.

If a masterbatch additive containing release agent (such as zinc stearate) is added to the material make sure that it has been added at the appropriate level. Verify that the letdown ratio is correct and thorough mixing is occurring. Do not take it for granted that the appropriate letdown ratio was used, especially if sticking and pin push start to occur "out of nowhere".

37.3.4.2 Material: Moisture Content

When hygroscopic materials are molded with excess moisture they often times tend to become stickier. This increased stickiness can lead to pin push due to the higher level of force required to eject the molded part.

See Chapter 9 on drying for more details.

37.3.4.3 Material: Fillers

The addition of fillers will have two positive effects on pin push:

1. Nucleating effect

   The nucleation impact on a semi-crystalline material will result in a part that has a more solidified wall stock, thus reducing the risk of pin push.

2. Stiffness of the part

   The addition of fillers will tend to provide a stiffer wall that is more resistant to pin push defects.

38 Read-through

38.1 Description

Read-through is a defect that creates a gloss difference on the class "A" surface of the molded part that mimics some detail from the backside of the part. Figure 38.1 shows an example for read-through.

Also known as: gloss, pin read, lifter read, wall stock transition

Mistaken identities: pin push, sink

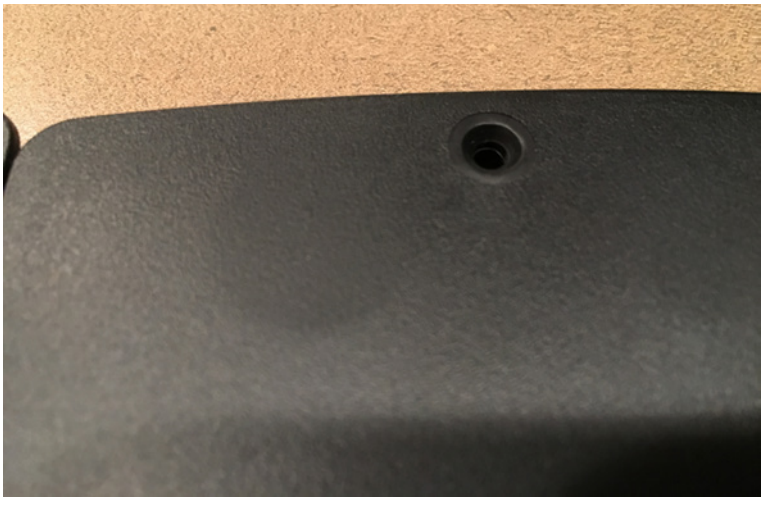


Figure 38.1 Read-through from ejector pin

38.2 Read-through Troubleshooting Chart

Table 38.1 shows the read-through troubleshooting chart.

Table 38.1 Read-through Troubleshooting Chart

Molding Process	Mold	Machine	Material
high second-stage pressure	component deflection	machine performance	
mold temperature	lifter spot		
	cooling		
	wall stock changes		

38.3 Read-through Troubleshooting

Read-through can be a challenge to troubleshoot because it is often a mold and process combination that leads to the problem. The root cause of read-through is usually either differential shrink or deflection of mold components. Determine what is located on the part behind the read-through: look for ejector pins, lifters, and wall stock changes. The Observation portion of STOP troubleshooting is especially important when dealing with read-through.

38.3.1 Read-through Troubleshooting Molding Process Issues

Potential process impact on read-through includes:

- High second-stage pressure
- Mold temperature

38.3.1.1 Molding Process: High Second-Stage Pressure

High second-stage pressure can lead to two main concerns:

1. High pressure can lead to mold component deflection. High cavity pressures may actually deflect ejector pins, lifter rods, and even ejector plates. See Section 38.3.2.1 for details on component deflection. Try reducing second-stage pressure to determine impact on read-through.
2. High second-stage pressure may lead to differential cavity pressures that may in turn lead to areas that read through due to differential shrink.

38.3.1.2 Molding Process: Mold Temperature

Uneven mold temperatures can lead to a difference in gloss levels on a molded part. If a mold component is not cooled well enough the results will be differential shrink. This differential cooling and differential shrink can lead to variation in gloss levels.

Ensure that the mold surface is running evenly. Surface temperature checks with a pyrometer will provide information on the actual running temperature of the mold. Do not rely on thermolator set temperature to determine mold temperature. Verify the surface temperature and part ejection temperature versus the documented process.

38.3.2 Read-through Troubleshooting Mold Issues

The true root cause for read-through is most often a mold-related concern such as:

- Component deflection
- Lifter spot
- Cooling
- Wall stock changes

38.3.2.1 Mold: Component Deflection

Mold component deflection is one of the most typical reasons for read-through. When a mold component deflects under molding pressures it will tend to spring back when the pressure drops, which can result in read-through.

Components that commonly experience deflection are ejector pins, lifter rods, cores, and ejector plates. The shape of the component will be what is normally seen in the read-through defect.

When it comes to mold components such as ejector pins, lifter rods, and cores the longer the unsupported length the more likely the component will bend under cavity pressure and then flex back against the plastic resulting in read-through on the part. Always make sure that there is adequate bearing surface and support to minimize the amount of unsupported length to avoid deflection.

To limit deflection of ejector plates it is necessary to have ejector plate stop buttons located behind the ejector plate to back up the area of deflection. If only the standard four-stop buttons are used there is opportunity for the mold components to flex the ejector plate, which will result in read-through. During mold design utilize adequate ejector plate stop buttons to limit ejector plate deflection.

Case Study: Lifter Detail Deflection

In this case, a filled polypropylene part was showing visually objectionable read-through at all lifter features. Initial troubleshooting presumed that the temperature of the lifters was the root cause of the problem. However, there were two indicators that the temperature of the components was not the root cause of the read-through. Firstly, the defect was visible on the first shot with a fully cooled mold and did not get worse, and secondly, changing mold temperature had no effect.

As a solution the tip of the lifter was supported with a pin behind the lifter head. This pin kept the tip of the lifter from flexing under cavity pressure and led to elimination of the read-through on this part.

38.3.2.2 Mold: Lifter Spot

If a lifter is not spotted such that it is bottomed in the lifter pocket, the lifter will be forced back by injection pressure, which will lead to reading of the lifter. A lifter should be well spotted to the bottom in the lifter pocket to minimize the amount of lifter movement when exposed to pressure from injection.

To check lifter spot the lifter and pocket can be cleaned and blued with die blue to inspect for contact between the pocket and lifter head.

38.3.2.3 Mold: Cooling

To limit differential shrink it is important to ensure uniform cooling across a mold. This often means that mold components such as slides and large lifters may need improved cooling strategies. To improve cooling of mold components use of alternate materials such as copper alloys can allow greater heat transfer. Whenever using copper alloys to improve cooling it is critical to get water lines to the component. If using a copper alloy without water lines to it the copper alloy will conduct the heat from the plastic efficiently but if the component is not cooled the result will be that the component runs hotter than the surrounding mold. Always provide cooling water to alternate materials such as copper alloys to provide an outlet for the heat that is transferred from the plastic. There are many ways to sneak small bubblers into mold details (see Chapter 14 on mold cooling).

Often times it is difficult to determine if a hot lifter, lifter movement, or deflection is the root cause of read-through. Experience has shown that the more typical cause is deflection or movement of the lifter rather than temperature. However, remember that both of these issues must be addressed during mold design.

Mold surface temperatures and part ejection temperatures will help determine where hot spots are occurring in the mold. If you are experiencing chronic mold cooling issues, have an experienced tooling person investigate options to creatively

add cooling to the mold. Adding cooling after the mold has been built will be much more expensive than if the concern was addressed during mold design, but it may be the only option to eliminate a production problem.

38.3.2.4 Mold: Wall Stock Changes

One of the fundamental design rules is to maintain nominal wall stock. If there is a change in wall stock the plastic will be packed and cooled at a different rate, which will result in differential shrinkage. During part design all efforts should be made to limit areas of varied wall stock. Thick areas of the part should be cored out to maintain nominal wall stock. If there is a spot on a part that cannot be cored out try to design in a gradual transition that will help limit the chances of read through that comes with abrupt wall stock transitions.

If a nominal wall stock change must occur on a part another factor that can help is designing in means to hide the read-through on the class "A" surface. Ways to hide read-through include design features that disguise the read-through such as style lines. Another way to minimize read-though concerns is use of a textured mold surface.

38.3.3 Read-through Troubleshooting Machine Issues

Machine performance should be verified (see Chapter 8).

38.3.4 Read-through Troubleshooting Material Issues

Read-through is a problem that can occur across all different types of materials. Viscosity is the main impact that material variation has on read-through. If material viscosity is varying over time, whether due to incoming material variation or moisture content, the cavity pressure will also vary, which leads to problems with variation in read-through.

39 Scuffs and Scratches

39.1 Description

Handling damage accounts for any defects that occur either as the part is removed from the mold or after it has been removed from the mold. Typical concerns would be scratches, scrapes, broken parts, or surface contamination. See Figure 39.1 for an example of handling damage.

Also known as: handling damage, scrapes, dings, breakage

Mistaken identities: splay, contamination, mold damage

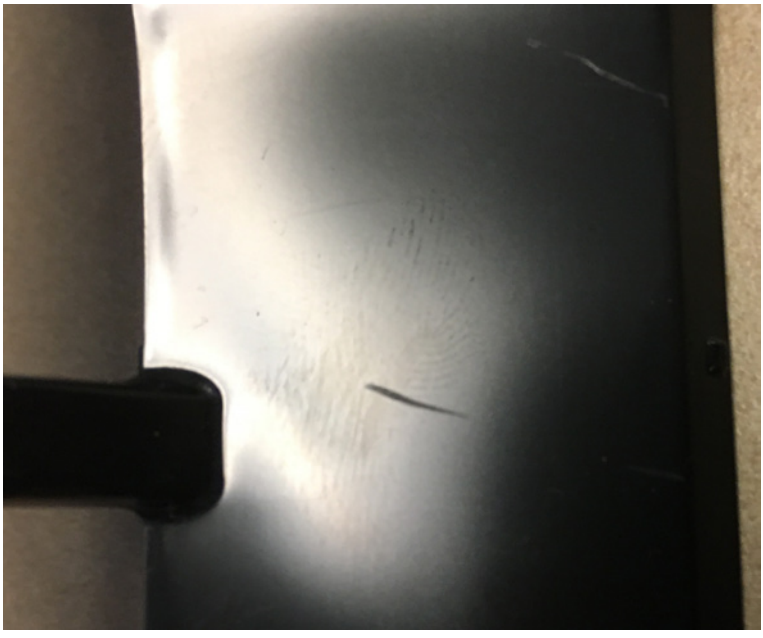


Figure 39.1 Surface scratch from handling

39.2 Scuffs and Scratches Troubleshooting Chart

Table 39.1 shows the scuffs and scratches troubleshooting chart.

Table 39.1 Scuffs and Scratches Troubleshooting Chart

Molding Process	Mold	Machine	Material
robot handling	texture	machine performance	material type
ejection issues	lack of ejection		
packing parts	parting line damage		

39.3 Scuffs and Scratches Troubleshooting

One of the keys to fixing handling damage is to correctly identify the problem. If the defect is a scratch but it is getting called splay, troubleshooting efforts will not follow a logical path. Using the STOP method is critical to resolving handling problems, especially the Observation step. Watch closely for opportunities for damage to occur.

39.3.1 Scuffs and Scratches Troubleshooting Molding Process Issues

For the discussion anything that occurs during the mold open sequence will be considered as a part of part handling:

- Robot handling
- Ejection issues
- Packing parts

39.3.1.1 Molding Process: Robot Handling

The use of a robot to remove the part from the mold usually helps prevent damage to the part. By avoiding ejecting the part into a drop chute the part is better protected.

However, the robot can create its own set of potential defects. One of the most common occurrences is when the end of arm tooling (EOAT) is not adjusted correctly and a piece of the framework for the EOAT hits the part leaving a scuff or scratch.

Another factor that can impact the robot and cause damage to the part is if grippers or nippers are misadjusted. A misadjusted nipper could be cutting off a section of the part at worst case. Sometimes with grippers there is a need to add some padding to prevent the metal of the gripper from scratching the part.

Watch the robot through its full cycle to determine if the part is contacting anything in the molding cell. There have been cases where the part hangs slightly too low and brushes against a guard resulting in scratches. Inspect the part on the EOAT to determine if anything is misadjusted and causing damage. Watch the robot to verify that the parts are not being dropped from excessive heights as damage may occur upon impact or the part may flip onto the class "A" surface. Also, parts dropped onto a conveyor can cause tabs and part detail to bend or break (see Figure 39.2).

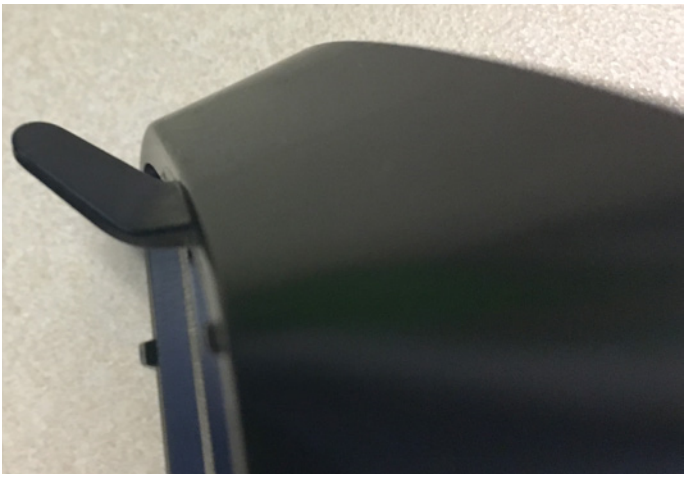


Figure 39.2 Bent tab from dropping on conveyor

Dirty or damaged vacuum cups can scratch a part surface. Examine the cups on the EOAT to determine if the cups are in good enough shape. One technique that can help identify if defects are coming from EOAT issues is to mark a part while it is on the EOAT with a magic marker by circling the vacuum cups. Any subsequent defects can be compared for location versus the marked part.

If the robot is picking parts that are very hot and the sprue is still molten, caution must be exercised to avoid allowing the sprue to touch the part and create a melted spot on the part. Watch carefully as the robot maneuvers the part and places the part and runner on the conveyor. If possible look for opportunities to drop the runner separately from the part to minimize risk of damage.

Watch the forward ejection stroke on the machine as well. If someone adjusted the forward stroke, the robot may be hitting the parts harder than previous runs. The robot pick position works in concert with the forward ejection stroke, so if one is changed the other probably needs to be changed as well.

39.3.1.2 Molding Process: Ejection Issues

A common way to run a molding machine is to eject the parts onto a chute or conveyor rather than robotically removing the parts from the mold. When parts are ejected onto a chute or conveyor they can bang into each resulting in handling damage. Normally this type of ejection should be reserved for noncosmetic parts. Keep in mind, however, that running without a robot can speed up cycles by reducing the mold open time. Running without a robot works very well with the right parts.

Sometimes when a part is ejected from the mold it may contact other areas of the mold and damage the part. One example of this is when parts are ejected and contact a leader pin, leaving grease on the part. Another example is when the parts are ejecting fast enough that they hit the cover half of the mold.

If the sprues are still hot enough they can cause a spot of remelted plastic on the part when they are dropped together in the chute or conveyor. An improvement to this may be adding a sprue picker to the operation to keep the parts and sprues separate. Improving the cooling to the sprue may also help the issue, as may downsizing the sprue. Adding a hot tip heated sprue bushing to the mold would be another option to eliminate this defect.

Care should be taken in deciding how the part should be removed from the mold. As mentioned above, cosmetic parts may need a robot to prevent damage. Whether parts are ejected into a chute, conveyor, or robot, the opportunities for handling damage must be assessed.

Watch for cases where a part is pulled back onto the core of the mold when the ejectors return. This will often cause distortion especially on molds with lifters. If the part is not consistently ejecting with one stroke, the mold should be evaluated for possible improvements.

39.3.1.3 Molding Process: Packing Parts

For the sake of consistency, packaging will be included in the molding process. Once the part has been successfully delivered to the machine operator there is another range of possibilities that can cause damage including:

- Operator damaging parts with jewelry or belt buckles
- Operator damaging the part with dirty hands
- Operator scratching part against work station

- Incorrect loading of pack
- Dropping parts on the floor
- Stacking parts on top of each other
- Poorly designed packaging that does not protect the parts
- Bulk packing parts that should be cell packed or layer packed
- Improper trimming by the operator

Thorough inspection of the part handling after it has left the equipment is an important step to diagnosing handling problems. Often times one of the above items will result in parts being damaged at the operator station.

Case Study: Scratching Parts

This example was a case where the first step in troubleshooting was understanding the true defect. This particular polypropylene part was experiencing high scrap numbers for splay. Using the Think step of the STOP process highlighted that splay is not usually a problem on unfilled polypropylenes. Using the Observation step showed that what was being scrapped as splay actually had scratches that were occurring as operators bumped the parts against a sharp corner on the work table. The corner was fixed and the "splay" problem was gone.

It is important to let just one piece flow through the operator work station on parts that are at risk for handling damage. When multiple parts are at the work station there is a much higher likelihood that parts will be stacked or banged against each other resulting in damage.

39.3.2 Scuffs and Scratches Troubleshooting Mold Issues

There are certain features in mold design that can lead to concerns with handling damage including:

- Texture
- Lack of ejection
- Parting line damage

39.3.2.1 Mold: Texture

When a mold is textured it is important to make sure that the texture is not too deep for the draft of the part because otherwise scuffing will occur. When inadequate draft is used with a given texture the peaks of the texture on the part will drag as the part is ejected, which will leave behind texture damage in the form of scuffing.

Texture on side walls must be accounted for during part design because adequate draft must be used for the chosen texture. A lack of draft will lead to a situation where the texture is "scrubbed" from the mold or reduced to accommodate part ejection without scuffing. Bear in mind that for every 0.001 in of texture depth the draft angle may need to increase by as much as 1°.

It is important to understand that parts that are molded with very low gloss levels (under 2.5) will tend to show marring very easily. A slight difference in gloss level can provide a part that provides a much more stable surface appearance.

Another detail to watch for is parting lines that have been rolled over creating an undercut. These undercuts in the steel will cause a drag or scuff when the part is ejected from the mold. A ruby stone can quickly clean up a rolled parting line. To detect a rolled parting line drag a fingernail across the suspect location; if there is a rolled edge your fingernail will catch on the edge.

39.3.2.2 Mold: Lack of Ejection

A mold must have enough ejector area to successfully remove the part from the mold. If there is not enough ejection area in a mold the part may be trapped on the mold. Lack of ejection area can lead to a variety of damage including pin push.

Whenever ejector pins are designed in a mold the best situation is pushing the part off the mold rather than pulling the part off the mold. In other words, the ejector pins, sleeves, and lifters should be pushing on the deepest areas of the part whenever possible. If the ejectors are pulling the part off the mold the chances of a rib or boss sticking is elevated.

Another area that can create damage is if a part sticks on a lifter or slide and tries to travel with the component during ejection. To avoid this problem ensure that components have an adequate draw polish for ease of movement. Another trick that can help with this problem is to add "donuts" to the ejector pins near the lifters. These "donuts" are rings that trap an ejector pin on the part; this will help hold the part steady so it does not travel with lifters.

Guided ejection is another aspect that is important relative to potential ejection damage. If parts are not ejected with the ejector plate moving forward in a square orientation there may be a chance for part damage to occur from sticking or scuffing. Guided ejection will help to keep the ejector plate square as it travels forward.

39.3.2.3 Mold: Parting Line Damage

If the parting line of the mold becomes rolled or has a burr it can cause a scratch or scuff on the molded part. When the scratch is in a consistent location examine the mold for damage that could be leading to the defect. Damaged parting lines need to be repaired to eliminate the defect.

39.3.3 Scuffs and Scratches Troubleshooting Machine Issues

If handling damage is being caused by a machine-related problem the most likely concern will be machine performance (see Chapter 8).

39.3.4 Scuffs and Scratches Troubleshooting Material Issues

The main thing to keep in mind with materials and handling issues is that some materials will scratch and mar easier than others. Softer surfaces can be more susceptible to handling damage. With soft materials, even separating the gate from the part may lead to distortion and damage of the part.

40 Short Shot

40.1 Description

A short shot is a part that has not been completely filled with plastic (see Figure 40.1).

Figure 40.1 Example of a typical short shot

40.2 Short Shot Troubleshooting Chart

Table 40.1 shows the short shot troubleshooting chart.

Table 40.1 Short Shot Troubleshooting Chart

Molding Process	Mold	Machine	Material
light fill only weight	venting	non-return valve	viscosity shift
pressure limited	cavity balance	wrong nozzle	low moisture content
low fill velocity	plugged gate or hot tip	worn barrel	inconsistent material feed
low second-stage pressure	stuck plastic	machine performance	unmelts
velocity to pressure transfer	low hot runner temperatures	nozzle leak	contamination
no cushion	leaking hot runner manifold		
no back pressure			
low melt temperature			
low mold temperature			

40.3 Short Shot Troubleshooting

When dealing with short shots focus on why there is a lack of plastic or plastic pressure at the location of the short shot. Use the STOP method to systematically work through the potential root causes.

40.3.1 Short Shot Troubleshooting Molding Process Issues

Potential molding process related causes are:

- Light fill only weight
- Pressure limited
- Low fill velocity
- Low second-stage pressure
- Velocity to pressure transfer
- No cushion

- No back pressure
- Low melt temperature
- Low mold temperature

40.3.1.1 Molding Process: Light Fill Only Weight

When using RJG Decoupled Molding® techniques the mold should be filled during first-stage injection to 95–98% full. If the mold is not filled far enough during first stage the mold may end up with a short shot.

Verify that the fill only weight matches the documented process. If the fill only weight is too light investigate potential causes, which can include:

- Shot size too small
- Transfer position too large
- Leaking nozzle
- Leaking non-return valve
- Leaking manifold
- Timing out on first-stage injection time

Before adjusting shot size or the transfer position, it is critical to determine that the plastic is not going somewhere where it does not belong. There are many cases of someone fixing a short shot by adding plastic to the shot size only to increase the amount of plastic leaking into the hot runner manifold (see Figure 40.7).

Fill only weight is critical to ensuring that the mold is filled reproducibly. If the fill only weight is too light the volumetric flow rate may have changed during mold filling.

40.3.1.2 Molding Process: Pressure Limited

A molding process is pressure limited when there is not enough pressure available during first-stage injection to achieve and maintain set velocity. A process can be pressure limited due to a number of factors including:

- Machine capability. The machine may not have enough pressure available.
- Peak injection pressure setting set too low. Make sure that this is set approximately 20% above the required peak first-stage pressure. This will allow for variation in viscosity.
- Plugged feed system. A piece of metal or other contaminate may be trapped in the nozzle tip or the hot runner manifold causing pressure to spike.
- Cold temperature somewhere often in the nozzle or hot runner
- High pressure drop through nozzle, runner gate, or part

If a process is pressure limited the first-stage velocity will decrease during mold fill. Figure 40.2 shows process monitoring curves for a pressure-limited process;

notice that the hydraulic curve flattens before transfer and the velocity curve starts to decrease as the machine becomes pressure limited.

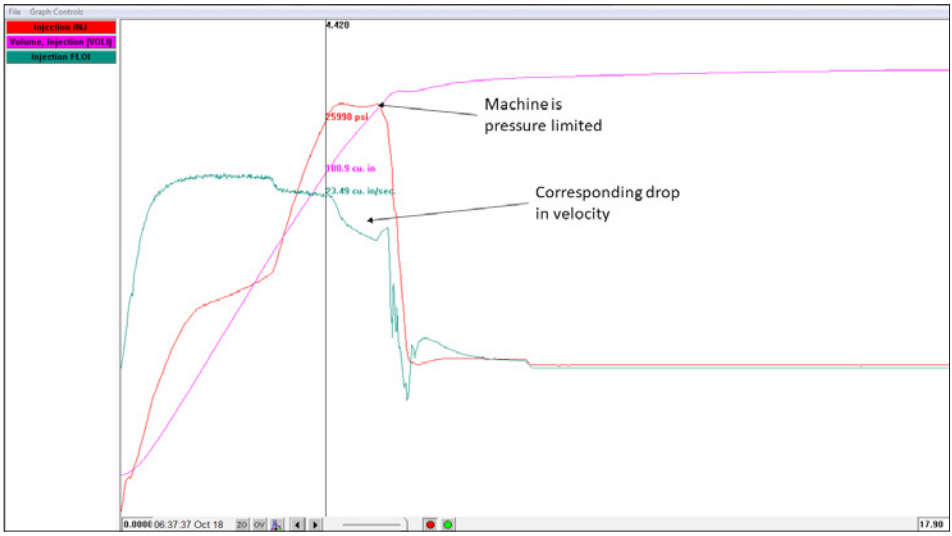


Figure 40.2 Process monitoring curves showing pressure-limited process

During process development make sure that the process is not pressure limited. Keep in mind that there must be an abundance of pressure (at least 10%) above the peak pressure required to fill the mold. If a mold is determined to be pressure limited during process development, effort should be made to determine what is causing the high pressure and address the issue.

If high first-stage pressure is a problem the first place to check for high pressure is the machine itself. Examine the nozzle and the nozzle tip to ensure that the correct tip orifice and style are on the machine. Also verify that the nozzle is not a mixing-style nozzle, which will add to the pressure drop through the nozzle.

The mold can also cause high pressure drops due to several factors including:

- Flow length to wall stock
- Gate style, size, and land
- Runner length, style, and size

The normal means of evaluating the cause of high pressure involves conducting a pressure drop study. A pressure drop stop is a series of short shots through different points of the mold and recording the peak pressure for each shot. Recent studies have shown that the traditional pressure drop study can be misleading [1].

40.3.1.3 Molding Process: Low Fill Velocity

If first-stage injection velocity is set too slow the mold will not fill completely. In most processes filling the mold fast is desirable because fast fill velocities help maintain a more consistent viscosity and limit the amount of pressure drop while filling the mold.

Verify that the machine is achieving the required fill time. If fill time is too long the actual first-stage velocity is likely too slow. If the fill time is not being achieved raise the velocity setting to achieve the documented fill time. Fill time and fill only weight are the normally used measures of actual volumetric flow rate. The fill velocity set point on the machine controller may not mean anything; for example, some machines use percent as the set point for velocity. Always rely on fill only weight and fill time to establish first-stage injection. Figure 40.3 shows data from process monitoring showing a reduction in fill velocity and the corresponding drop in cavity pressure. This reduction in cavity pressure can result in short shots.

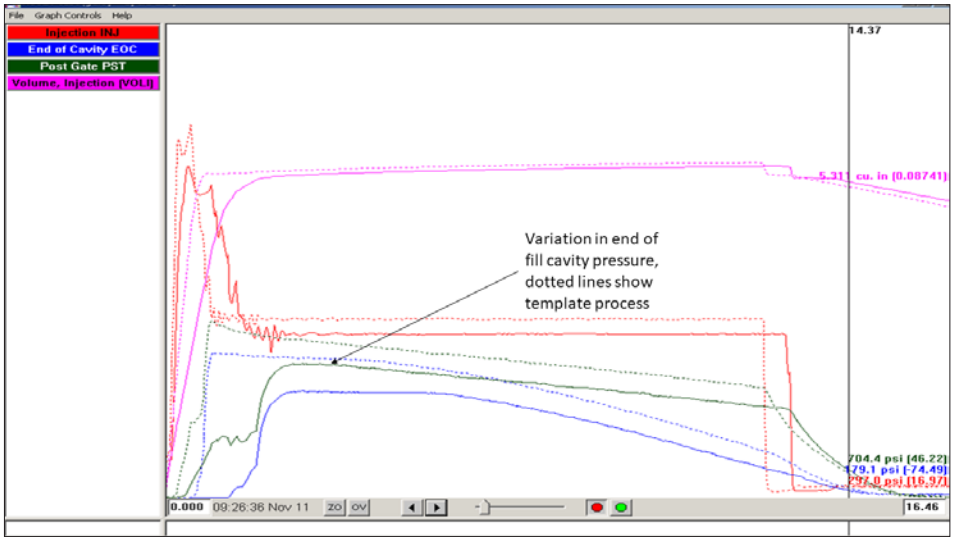


Figure 40.3 Process monitoring data showing template values (dotted lines) versus actual process curves after a fill speed reduction

There are cases when ribs or other details close to a gate will not fill at higher first-stage velocities. In these cases, the melt front is flowing past the rib following the path of least resistance and the rib detail has to backfill as the pressure in the cavity increases. This case can be more of a risk on ribs that are perpendicular to the melt flow. In these cases, slowing down first-stage fill velocity may help the rib from shorting out. Before slowing down the fill velocity, check for proper venting of the rib or mold detail.

Avoid processing around mold problems. In some cases, to limit flash, the fill velocity or transfer position will be modified, which in turn leads to short shots. Many times processing around one problem creates another defect.

40.3.1.4 Molding Process: Second-Stage Pressure Low

If second-stage pressure is too low the mold will not finish filling after transferring from first stage. Running an RJG Decoupled® process means that the mold will be only 95–98% full with no second-stage pressure applied. As pressure is applied to the plastic the remaining 2–5% of the cavity will fill and the part will be packed out. Some details in the mold such as thin ribs will require enough cavity pressure to force the plastic into the detail, but if cavity pressure is not high enough the part will be a short shot.

Compare the actual second-stage pressure to the documented process. If cavity pressure transducers are available verify that the cavity pressure is meeting the template value. Account for intensification ratio when molding on a hydraulic machine. If the pressure is low determine why; it may be a simple incorrect setting or may be a machine problem.

40.3.1.5 Molding Process: Velocity to Pressure Transfer

The transition between first-stage velocity and second-stage pressure can result in short shots. Depending on how fast the machine reacts and reaches set pressure the part may not fill out. Figure 40.4 shows process monitoring data for a bad velocity to pressure (V to P) response. Notice how all the three cavity pressure curves follow the dip in pressure that the machine experiences during transfer. For more details on velocity to pressure response see Chapter 8 on machine performance.

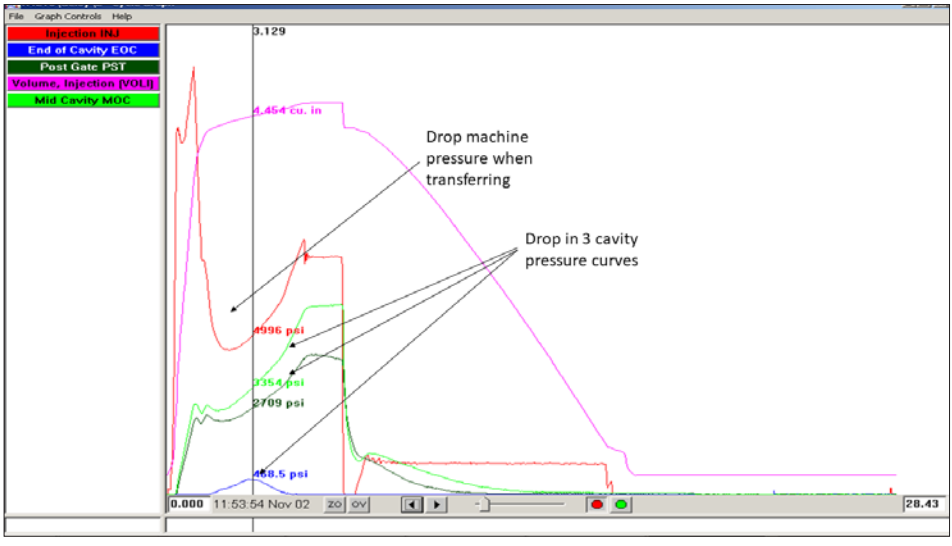


Figure 40.4 Process monitoring curves showing poor velocity to pressure response

40.3.1.6 Molding Process: No Cushion

The cushion is the plastic left in front of the screw tip after the mold has been filled. If there is no cushion the machine cannot transfer pressure to the plastic in the mold.

Verify that the machine is holding a cushion. A cushion reaching zero is an indicator that either the shot size is too small or plastic is leaking somewhere. Plastic may be leaking in the following areas:

- Past the non-return valve
- Between the nozzle tip and the sprue bushing
- Inside the hot runner manifold
- Through the mold parting line

If there is no cushion on the machine and there is no material leaking, the shot size should be re-established by checking the fill only weight. Usually shot size and transfer position will need to be adjusted.

Cushion monitoring is an excellent value for process monitoring. Alarms can be set up if the cushion drops below an established limit.

40.3.1.7 Molding Process: No Back Pressure

This is a problem that most often occurs after purging a machine. Often the screw will not recover due to the back pressure when purging the material. A common way to overcome this is to reduce the back pressure. If the back pressure is dropped to zero or a very low level the shot will be inconsistent and lacking in density, which can result in short shots.

If back pressure is reduced to purge a machine, double check to be sure that the pressure is returned to the required setting as this is easy to forget. Another option is to use decompression to help pull the screw back to shot size rather than reducing the back pressure.

Case Study: No Back Pressure

In this case the process technician was fighting with inconsistent short shots. While troubleshooting, the process technician realized that the cushion was bottoming out. To combat the lack of cushion the process technician tried adding shot and adjusting transfer while checking the fill only shot. He continued to struggle to get a consistent shot and called for process engineering support. Upon arriving at the machine, the process engineer used STOP methodology and a 4M basic 8 check to quickly determine that the process was running with zero back pressure. Adjusting the back pressure to the documented pressure allowed the process to be reset at the established settings.

▶

The process technician had reduced the back pressure while purging to allow the screw to recover a full shot. After building a shot and starting the machine on cycle the process technician forgot to reset the back pressure.

This case was an excellent reminder to check the process while on cycle and always STOP to think through the potential root causes.

40.3.1.8 Molding Process: Low Melt Temperature

Low melt temperatures often raise the viscosity of the melt resulting in material that flows less easily. High viscosity material will not fill part details as easily and will lead to an increased pressure drop that may cause short shots.

Verify that the melt temperature matches the documented process and is appropriate for the material. Increasing melt temperature is often done to help a difficult-to-fill mold. Keep in mind that added melt temperature increases cooling time and may lead to material degradation. Use a melt temperature probe or a thermal imaging camera to determine melt temperature, and raise it if it is low.

40.3.1.9 Molding Process: Mold Temperature

Low mold temperatures can impact the ability of the plastic to fill the mold. Cooler mold temperatures will cause a thicker frozen layer and potentially restrict flow in the mold. As plastic flows through a mold the mold surface temperature will rise (see Figure 40.5). If a mold has areas that are difficult to fill initial startup shots may not fill due to a low mold surface temperature.

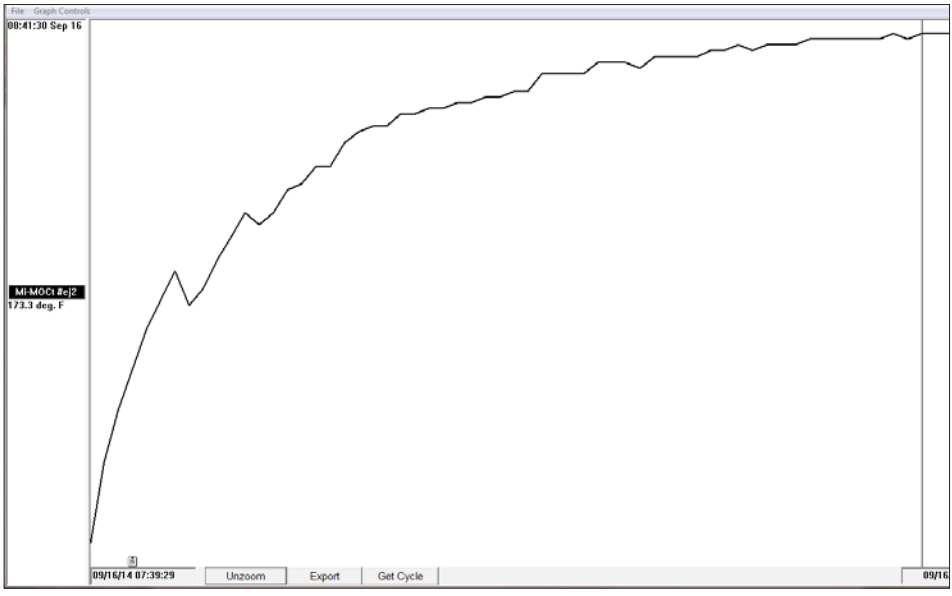


Figure 40.5 Mold surface temperature rise over 50 consecutive shots

Mold surface temperatures should be measured and documented after running on process. This will provide data of what the running mold surface temperature is. The set point of the mold temperature controller is not what the mold temperature will be. Also utilize thermal imaging of part out temperatures after the part has been ejected.

40.3.2 Short Shot Troubleshooting Mold Issues

There are many mold-related concerns that can lead to short shots including:

- Venting
- Cavity balance
- Plugged gate or hot tip
- Stuck plastic
- Hot runner temperatures
- Leaking hot runner manifold

40.3.2.1 Mold: Venting

One of the biggest mold-related causes of short shots is venting. If venting is lacking in a mold, trapped gas can lead to short shots. Trapped gas can result in either short shots or burns; both can indicate venting problems.

When troubleshooting short shots always make sure that the mold surface is clean and the vents are open. A dirty mold surface will restrict venting ability. On materials that are more prone to plate out, the vents will plug easier and must have even more focus.

For more details on venting see Chapter 7.

40.3.2.2 Mold: Cavity Balance

When a mold is unbalanced between cavities there will be an opportunity to have specific cavities short out while others are full. A multicavity mold should be balanced for fill within 3% to provide equal process conditions to all cavities. If a mold is unbalanced there will be a stop/start effect during filling, and as pressure equalizes, the plastic will follow the path of least resistance. When the first cavities fill, the lagging cavities will continue to fill, which can in turn lead to hesitation between filling and packing of the various cavities.

Figure 40.6 shows the normal imbalance of an eight-cavity mold caused by shear rates. Notice how the inner cavities are farther ahead than the outer cavities. For more details see Chapter 12 on cavity balance.

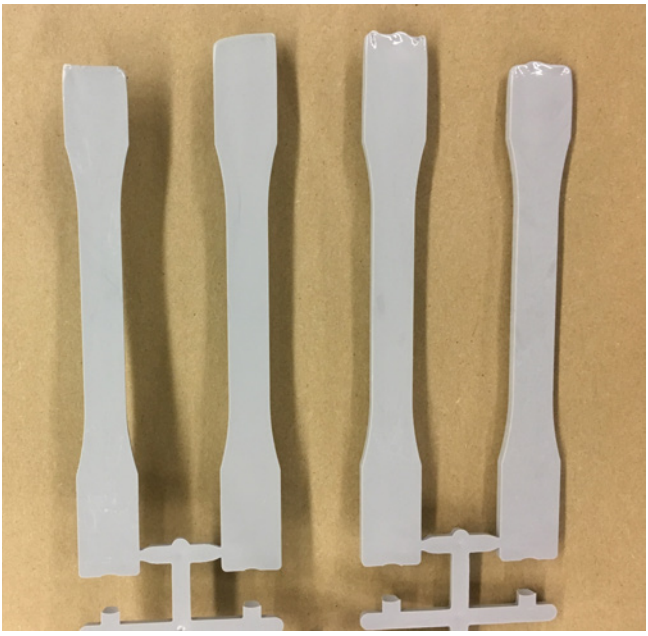


Figure 40.6 Typical eight-cavity shear-induced imbalance

40.3.2.3 Mold: Plugged Gate or Hot Tip

If foreign material becomes plugged in a gate or hot tip the flow to the cavity will be restricted. The result of a plugged gate or hot tip is often a short shot due to lack of material flow. Metal is the most common material that ends up plugging tips; however, contamination from carbon buildup or a higher melt temperature material can also lead to plugged tips.

On a mold with multiple tips either for multiple cavities or multiple gates on a single part a good check is to run a fill only shot and evaluate if all drops/tips are providing similar flow. If one drop/tip is lagging in volume flowing through it a plugged tip may be the root cause.

If a plugged tip is suspected a qualified mold repair technician should clean out the drop and tip combination. Beware of technicians on the floor trying to clear out a tip especially if it is a low vestige tip. The low vestige tip has a metal spreader inside the tip and there have been cases where a process technician tried to remove the metal in the tip, which caused damage to the spreader.

Case Study: "There's what in the hot tip?"

In this case the part had eight direct hot drops. Shorts were starting to show in a location that pointed to two of the gates having some kind of issue. Short shots showed that indeed there was no flow coming from these two gates. The temperatures of the drops were checked and everything looked OK, so the mold was pulled from the machine and taken to the tool room.

Examination of the mold in the tool room showed that the drops seemed to be coming to temperature without issue. An attempt to insert a thin probe into the drop failed due to hitting something hard in the tip. The tool room decided to drill out the drop and eventually hit the hard spot. They were able to get the foreign material out of the drop and examination of the material showed it was stone!

Discussions with the material supplier led to the possible root cause that the transfer lines used to load the material from the tanker truck to the silo was dropped in the parking lot and picked up a piece of gravel that then was loaded into the silo and made it all the way through the system until it reached the drops where it could not get any further.

If the suspected culprit is contamination from a higher melt temperature material, raising the drop and tip temperature may allow the material to melt and flow through the restriction of the tip.

40.3.2.4 Mold: Stuck Plastic or Buildup

Sometimes a part will have some detail such as a rib missing and look like a short shot. Before trying to resolve a short shot problem check to make sure that plastic is not stuck in the mold detail. Sometimes the detail breaks off due to sticking and stays trapped in the mold creating parts with the detail missing.

If this happens on a mold it may well indicate that the detail in the mold is lacking in polish and more prone to stick as a result. Inspect the rib for electrical discharge machining (EDM) marks or poor polish. Sometimes cleaning up the EDM marks in a rib will correct this problem from reoccurring.

Another factor that can magnify this problem is a mold started up with degraded plastic, which will be more brittle and sticky. Thorough purging is necessary when molding materials that are prone to degradation. Also start a mold using a fill only shot rather than using full pack pressure during startup.

Materials that are very gassy and not vented well can actually build up in deep mold details and leave what looks like a short shot. It may be necessary to clean out the buildup and add venting to prevent this problem from reoccurring.

40.3.2.5 Mold: Hot Runner Temperature

Low hot runner temperature can result in an increased pressure drop as the mold fills. Drop and tip temperature seem to have the biggest impact on flow of the plastic.

Check all hot runner zones and determine that everything is set correctly. Also check to make sure that all zones are drawing amperage and the thermocouples are reading correctly. If the hot runner controller is set correctly but the zone does not want to heat up check the following items:

1. Hot runner cables

   Verify that the correct number of zones are connected properly, and check for pins not pushed in.

2. Hot runner plugs on mold

   Verify that plugs are not damaged and cables are connected tightly.

3. Try another cable

   If another cable fixes the problem check the bad cable for continuity from pin to pin on the plugs.

4. After verification of the process the mold heaters and thermocouples will need to be checked.

A hot runner that is overheating due to a malfunction can lead to degraded material, which can in turn lead to short shots. Verify that the thermocouple is installed correctly and reading accurately. Also verify that the hot runner zones in question are providing accurate heating and not overriding set temperatures.

40.3.2.6 Mold: Leaking Hot Runner Manifold

If plastic is not making its way into the cavity one place where it can leak is within the hot runner manifold. Damage to the manifold can result in a plastic leak (see Figure 40.7). This is a perfect example of why the STOP mentality is so important: if the manifold is leaking and the technician adds shot size all they are doing is compounding the problem (a good lesson for a technician who does this is cleaning the plastic from the manifold). Such problems can become expensive and time-consuming repairs that always seem to occur when the mold is behind on production. Catching this problem early through cushion monitoring may reduce the amount of damage within the manifold and make for an easier repair.

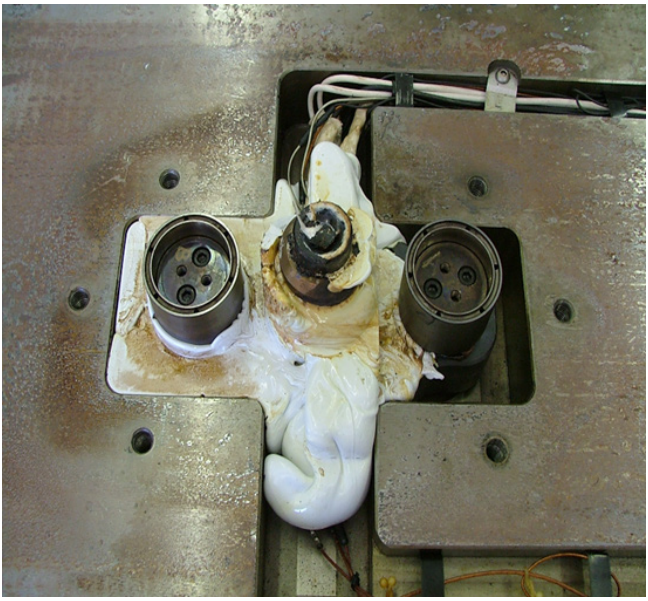


Figure 40.7 Leaking hot runner manifold

A common reason for leaking hot runner manifolds is cold starts. A hot runner must be allowed to achieve temperature and soak to ensure that all of the plastic in the manifold is melted and will move when a shot is injected through the mold.

There are cases of leaking manifolds that are severe enough that the mold is smoking due to burning plastic. Early detection is vital to dealing with this problem.

40.3.3 Short Shot Troubleshooting Machine Issues

Many machine concerns can contribute to short shots including:

- Non-return valve leaking
- Wrong nozzle
- Worn barrel
- Machine performance
- Nozzle leak

40.3.3.1 Machine: Non-return Valve Leaking

The job of a non-return valve on the screw tip is to keep material from flowing backwards during injection. If a non-return valve becomes worn or damaged the valve will not seal properly and plastic will squeeze backwards through the non-return valve. Figure 40.8 shows a typical sliding-ring-style non-return valve and Figure 40.9 shows a ball-style non-return valve.

Figure 40.8 Sliding ring check valve

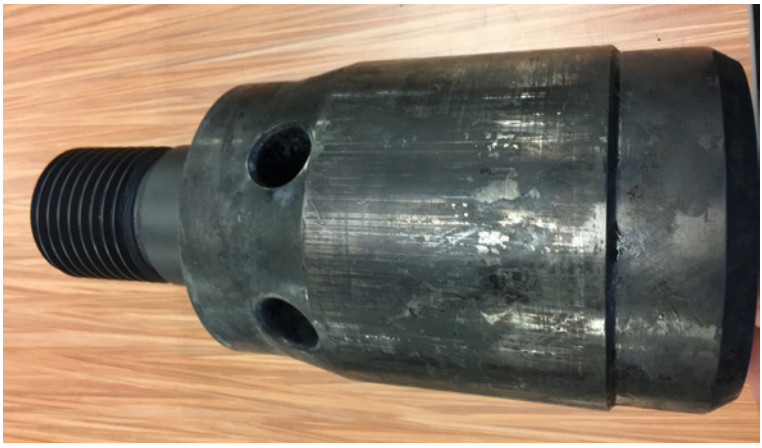


Figure 40.9 Ball-style check valve

With a leaking non-return valve the cushion value on the machine will either show a large amount of shot to shot variation or will consistently reach zero. Another clue that the non-return valve is leaking is to watch for the screw to rotate during injection because this is a sign that backflow is rotating the screw. Sometimes the cushion may not be reaching zero because the transfer position was large enough not to bottom the screw out. Adding second stage time will highlight if the screw stops moving or continues to travel until it bottoms out.

40.3.3.2 Machine: Wrong Nozzle

There are many choices for nozzles and nozzle tips that can be used on a molding machine. Each tip choice can impact the pressure on the machine. Nozzle concerns include:

1. Nozzle type. A mixing nozzle creates much greater flow resistance than a standard free flow nozzle. Mixing nozzles have static mixing elements inside them that are intended to improve the color mixing of the plastic. Mixing nozzles are really a Band-Aid to compensate for poor mixing as the material travels through the screw. As with many things there is a trade-off to the improved mixing that the mixing nozzle provides: negative aspects include increased pressure drop and potential material degradation.
2. Length of nozzle or nozzle tip. Establish the habit of using the nozzle and nozzle tip of the shortest length possible. The normal need for increased nozzle length is due to a hot runner with a recessed sprue. If the mold does not require an extended nozzle then do not use it. Longer nozzles and nozzle tips can impact pressure drop and will often have issues with proper heating. An example would be using an 8-in nozzle with a 4-in heater band; the result will be impacted by location of the band on the nozzle especially in relation to the thermocouple.
3. Nozzle tip style. The three most common are the general-purpose (GP), nylon, and full taper tip. There are also mixing tips on the market. The different tip styles will produce different pressure drops, different shear effects, and risks of contamination. A general-purpose nozzle has dead spots where the nozzle transitions into the nozzle orifice and can lead to contamination.
4. Nozzle tip orifice. The orifice of the nozzle tip should match the orifice of the sprue bushing on the mold. Normally a nozzle tip orifice that is 1/32 in smaller than the sprue orifice gives the best results.

During process development document the nozzle and all specifics including length, orifice, and style. All of these specifics should then be verified during every following mold run, because they are some of the details of molding that can lead to a great deal of problems. In molding it is important to “sweat the small stuff” because really there is no small stuff when replicating a process.

40.3.3.3 Machine: Worn Barrel

Sometimes what appears to be a defective non-return valve is actually a worn barrel. To determine if the problem is the non-return valve or the barrel try moving the shot size and transfer position back 1 in and running a shot. If the machine holds a steady cushion at the larger setting it is an indication that the machine has a worn spot in the barrel.

Use of glass-filled materials will lead to increased barrel wear. There are cases where glass-filled material running in a low hardness barrel lead to a worn-out barrel in less than six months. Barrel selection for glass-filled materials should lean toward bimetallic abrasion-resistant materials up to and including bimetallic carbide barrels and CPM-10 V high wear resistant barrels. For proper barrel selection determine what material will be run and choose the proper hardness.

Barrel diameters can be measured with a cylinder bore gauge to evaluate barrel wear. Screw diameters can be measured with an outside micrometer and a parallel bar that spans two flights of the screw.

40.3.3.4 Machine: Machine Performance

The machine must be capable of achieving the established set point of all setup parameters. Verification of machine performance is an important part of injection molding troubleshooting.

See Chapter 8 on machine performance for more details.

40.3.3.5 Machine: Nozzle Leak

If plastic leaks at the machine nozzle the ability of the process to provide consistent cavity pressure will be compromised. The main areas to verify for leaks are as follows:

1. Nozzle tip to sprue bushing. If the either the nozzle tip or the radius on the sprue bushing is damaged the plastic can leak through the gap. Another concern is the use of a nozzle tip with the wrong radius, which will create a gap at the contact between the nozzle tip and sprue bushing radius. See Figure 40.10 for an example of a nozzle to sprue plastic leak.
2. Before bringing the injection unit forward ensure that all leaking plastic has been removed from the nozzle tip and the sprue, and if using a hot runner make sure that the sprue bushing seat is clean. Repeated crushing of plastic at this interface will cause damage over time resulting in a leak.
3. If any of the end components of the machine are not tightened correctly a leak may occur. Possible leak areas could be between the nozzle tip and the nozzle, between the nozzle and the adapter, between the adapter and the end cap, and between the end cap and the barrel. When installing any of these components make sure that the seat surfaces are clean (including no melted plastic). Also make sure that all components are adequately torqued.

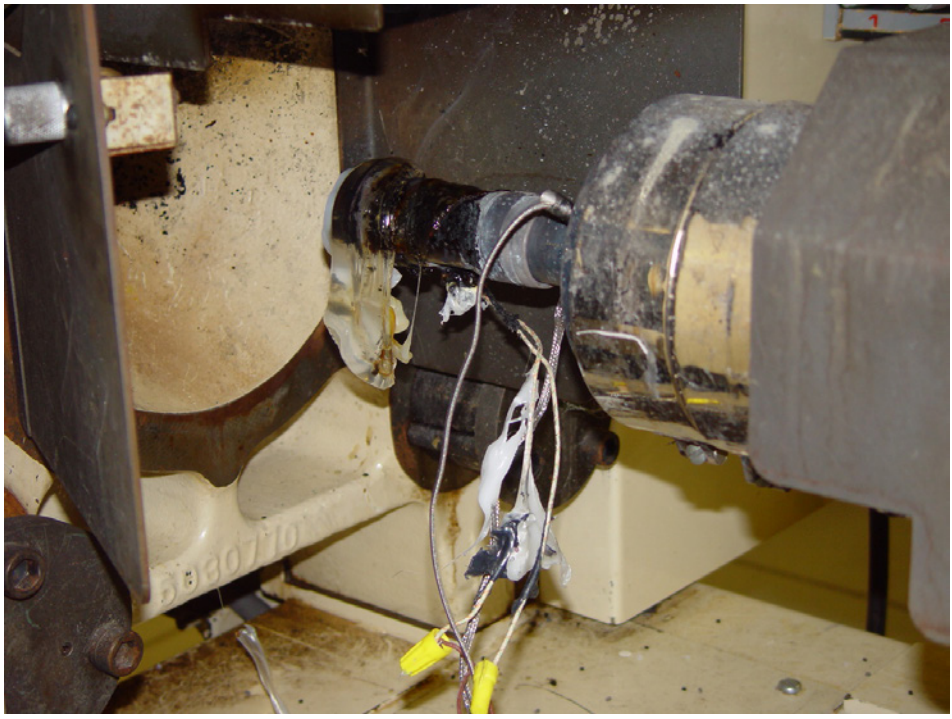


Figure 40.10 Example of nozzle to sprue leak

40.3.4 Short Shot Troubleshooting Material Issues

The material can cause short shots due to several issues including:

- Viscosity shift
- Moisture content
- Inconsistent material feed
- Unmelts
- Contamination

40.3.4.1 Material: Viscosity Shift

Viscosity is a measure of the resistance of a material to flow. As viscosity increases the material will not flow as easily. If material viscosity increases two main concerns will arise:

1. High viscosity material may not flow into difficult to fill areas of the mold.
2. Higher viscosity will lead to increased pressure drop across the mold resulting in lower cavity pressure at end of fill locations.

All materials will have some viscosity variation over time. If using wide-specification materials the variation will be much larger. Viscosity variation is one of the critical reasons a process needs abundant pressure for repeatable fill times. If the process is close to being pressure limited and the viscosity increases the process may become pressure limited and have a short shot.

If short shots start immediately after changing to another lot of material, a viscosity shift is most likely the cause. Keep in mind that a robust process should be capable of absorbing incoming viscosity changes. One way of validating viscosity impact on a process is to make sure that process development is conducted with multiple lots of material. If a process cannot produce quality parts through the range of viscosity variation, instrumenting the mold to run a Decoupled III® process may be the best decision. An RJG Decoupled III® process uses a cavity pressure transducer to transfer from the pack stage to the hold stage; this will help ensure that the mold has been packed to the same level. Also, through use of end of fill cavity pressure transducers the process can easily be monitored for short shots and set up to automatically reject suspect shots.

40.3.4.2 Material: Moisture Content

Moisture will decrease the viscosity of nylon. If process development is conducted using wetter nylon the mold will fill easier than with dry nylon. Dry nylon may impact the viscosity enough to cause short shots and even create a pressure-limited condition. Try to ensure consistent levels of moisture when molding nylon.

For details on material drying see Chapter 9.

40.3.4.3 Material: Inconsistent Material Feed

If the material is not being fed consistently to the machine short shots can result. As the screw is recovering a lack of feed can starve the screw and lead to inconsistent shots and possibly short shots.

If material is being fed from a container such as a gaylord it is important to ensure that the feed hose is consistently pulling material. A gaylord tipper may help with material feed.

Some materials such as long glass-filled materials and some regrind will be difficult to feed. In some cases a vibrator may need to be added to ensure that the feed throat does not bridge over with pellets.

When using central material feed systems make sure that there is adequate timing for all hoppers to load consistently. In some cases the feed system may be timed incorrectly, which can result in machines being starved of material between load cycles. Check the hoppers to ensure that material is being loaded in time to keep up with machines recovery rate. Feed times to the machines may need to be adjusted to allow adequate feed to all machines in the system.

40.3.4.4 Material: Unmelts

Unmelts can lead to short shots due to the potential to block gates. This is a bigger concern on a multicavity mold because the unmelts will impact cavity balance. An unmelt is an unmelted pellet that has made it through the barrel. This issue seems to be most common when running acetal materials, but may occur with other semi-crystalline materials. Unmelts will often hang in a gate until the pressure is high enough to force them through the gate; often leading to a short shot. An unmelt can also impact the seating of the non-return valve and lead to inconsistent fill weights.

Unmelt Test

1. After running the molding process for a minimum of 10 minutes, back the barrel away from the mold and purge out an air shot.
2. Immediately recover another shot and purge it out.
3. Continue to recover and purge shots until unmelted pellets appear in the purge. If this occurs after three shots or fewer the quality of the melt is not good, and the odds of unmelt problems will be high.

40.3.4.5 Material: Contamination

If the material becomes contaminated with foreign material a short shot can result. If the foreign contamination moves through the melt stream as a solid it may become trapped in a gate and lead to a short shot.

Check the material for any potential contamination. Review Chapter 23 on contamination for additional details.

References

[1] Beaumont, J., *Questioning the Science in Injection Molding: What are you Actually Getting in a "Pressure Drop Study"* Originally published in SPE Plastics Engineering Magazine; http://www.beaumontinc.com/questioning-science-injection-molding-actually-getting-pressure-loss-study/

41 Sink

41.1 Description

Sink describes areas of a part where the surface is not flush as intended (see Figure 41.1). There is a depression on the surface that is objectionable. Sinks and voids are interrelated because a void is really a sink that does not affect the surface of the part.

Also known as:

Mistaken identity: read-through

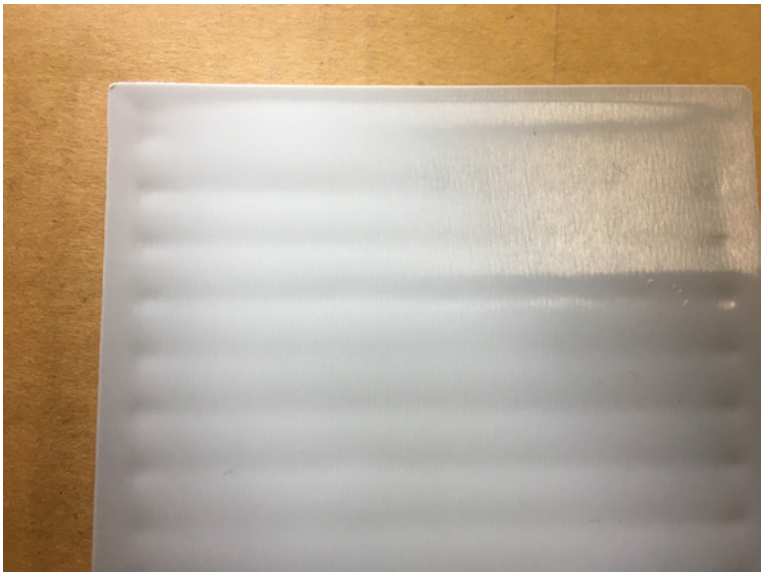


Figure 41.1 Sink

41.2 Sink Troubleshooting Chart

Table 41.1 shows the sink troubleshooting chart.

Table 41.1 Sink Troubleshooting Chart

Molding Process	Mold	Machine	Material
low second-stage pressure	plugged cooling lines	machine performance	viscosity shift
short second-stage time	poor cooling design	velocity to pressure transition	nucleating agent
low fill only weight	low/high hot runner temperatures	inconsistent non-return valve	blowing agent
lacking cushion	valve gates not actuating	nozzle tip type/style	
slow fill velocity	wall stock issues	gas assist not working	
high mold temperature	cavity balance		
high or low melt temperature	gate size, location, and quantity		
	pull sink		

41.3 Sink Troubleshooting

Sink can be a very difficult defect to correct if the part was designed incorrectly. Due to the shrinkage that occurs in plastic during the molding process thick wall sections will have a tendency to have sinks. Wall stock recommendations as well as rib to wall guidelines must be followed during the design of the part. Also, design rules such as not gating thin to thick must be followed to minimize sink. If the processor is left trying to process out sink they will only achieve so much; the laws of physics still apply.

The following troubleshooting guidelines are based on improving sink within the constraints that the part design produces.

41.3.1 Sink Troubleshooting Molding Process Issues

There are many factors in the molding process that will lead to increased amounts of sink on a molded part including:

- Low second-stage pressure

- Short second-stage time
- Low fill only weight
- Lacking cushion
- Slow fill velocity
- High mold temperature
- High or low melt temperature

41.3.1.1 Molding Process: Low Second-Stage Pressure

The pressure applied during the second stage of injection packs out the cavity and compensates for plastic shrink. Enough pressure must be applied to the plastic in the cavity to eliminate sink in the part. Many people will use rules of thumb for setting second-stage pressure but the best plan is to let the part dictate what is required based on cosmetics and dimensions.

Verify that second-stage pressure is set to the documented process. Double check that someone did not add a pressure profile to the machine controller. If the pressure is set incorrectly the process should be adjusted to the specified set point.

On machines that have pack or hold volume, speed, or velocity settings make sure that the setting is large enough to allow the machine to reach the set pressure. If volume, speed, or velocity is set too low on second stage the machine will struggle to reach the set pressure.

Watch for cases where second-stage pressure has been adjusted to process around a tooling issue. An example is someone adjusting second-stage pressure down because the mold has parting line damage, but the lowered pressure in turn produces sink. Fix the problem with the mold rather than trying to process around the damage.

41.3.1.2 Molding Process: Short Second-Stage Time

Cavity pressure in a mold is built with second-stage pressure and maintained with second-stage time. In the majority of cases second-stage time will be set to be just longer than the required time for gate seal (gate freeze), which is the point the plastic in the gate has solidified enough to prevent plastic in the cavity from flowing back out of the cavity.

Determine if the mold should run with gate seal. If a gate seal study has not been completed then take the time to verify gate seal. If a gate seal study has been completed compare the set second-stage time to what is required for gate seal. If necessary, increase second-stage time to ensure gate seal (if gate seal is required).

Lack of gate seal will often lead to sink near the gate and many times directly at the gate. However, running with second-stage time set too short can lead to sink throughout the part. If the mold has a cavity pressure transducer it is possible to

see the cavity pressure drop at the end of second stage indicating a sudden change in cavity pressure. Figure 41.2 shows eDART® curves for a process running without gate seal. Notice how the post gate cavity pressure curves drop at the same time as the machine pressure curve at the end of second stage.

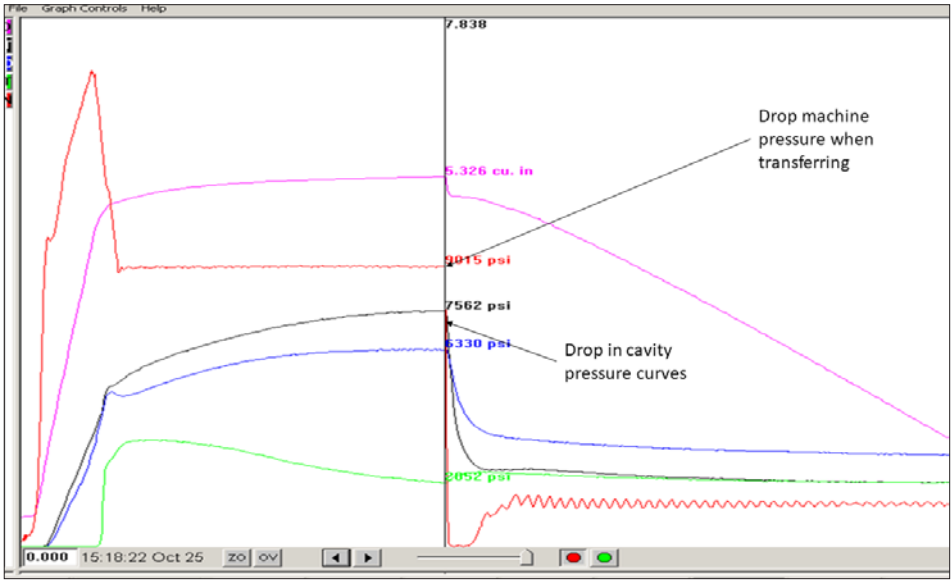


Figure 41.2 eDART® graph showing process without gate seal

Molding machines have a variety of names for the second stage including pack, hold, or just second stage. It is vital to understand how the machine is reacting to the various set points and utilize the variables appropriately. Often times on machines with pack and hold settings only hold will be utilized. Watch the pressure curves or pressure gauge to determine how the machine reacts to second-stage set points.

41.3.1.3 Molding Process: Low Fill Only Weight

When running using RJG Decoupled Molding® it is critical to have a consistent 95–98% fill only weight. Fill only weight should be documented for every mold.

To check the fill only weight, remove second-stage pressure and volume on the machine controller. The result should be a 95–98% full part. Next, weigh the short shot and compare to the documented fill only weight. If the fill only weight is too low adjustments should be made to achieve the desired fill only weight. Go through the following steps:

1. Verify that the machine is running with an adequate cushion (minimum of approximately 0.25 in).
2. If machine is holding a cushion, try reducing the transfer position.
3. If machine is not holding a cushion see Section 41.3.1.4.
4. Adjust shot if lack of cushion is due to shot size.

41.3.1.4 Molding Process: Lacking Cushion

To transfer pressure from the machine to the cavity the machine must have a cushion of plastic in front of the screw tip. Cushion is a valuable variable to document and track. If the machine is not holding a cushion, determine where the plastic is going. Possible areas of leaking are:

- Past non-return valve
- Between nozzle tip and sprue bushing
- In hot runner manifold

Wear or damage to the barrel, non-return valve, nozzle tip, or sprue bushing can lead to material leaking. Also hot runner manifolds can become cracked and leak plastic (see Figure 41.3). If the machine is not holding a cushion because of a leak **do not keep adding shot size!** If there is a leak the damage to the mold or machine will be costly and time consuming to repair. To dig the plastic out of the manifold would be an excellent learning experience in this case for the person who filled it.

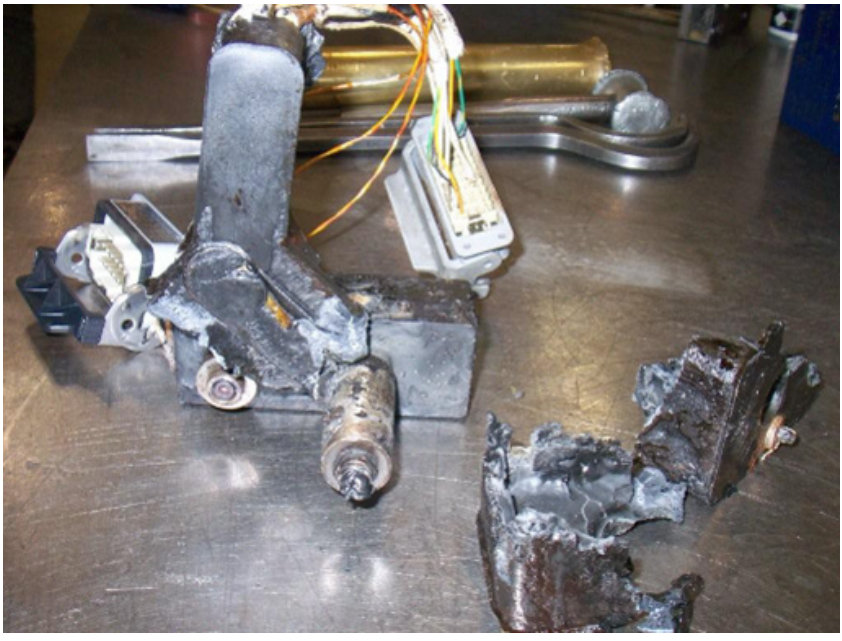


Figure 41.3 Remains of plastic that leaked in a hot runner manifold

If a non-return valve is leaking there will be some things to check:

- Make sure that the decompression is set correctly.
- Watch the screw during injection because if the non-return valve is leaking the screw will rotate during injection.
- Inject forward against the mold with a part and sprue in it. If the screw travels forward even though the plastic has nowhere to go the non-return valve could be leaking.
- Try moving the shot size and transfer position back 1 in and running a shot. If the shot holds at the larger transfer position it may indicate a worn spot in the barrel rather than a worn non-return valve.

41.3.1.5 Molding Process: Slow Fill Velocity

When a mold is filled with slow velocity during first stage the viscosity of the material will be higher leading to a larger pressure drop across the cavity. This increase in viscosity may reduce the ability to pack out the part.

The fill time should match the documented fill time. If the fill time is not matching the setup sheet the first-stage velocity should be adjusted accordingly. A repeatable fill time is one of the critical process requirements. If the first-stage velocity has been reduced due to a tooling issue such as burning, fix the mold.

41.3.1.6 Molding Process: High Mold Temperature

A high mold temperature will impact the cooling rate of the plastic. Hotter molds have a tendency to produce more shrink in the plastic which in turn can lead to additional sink.

Ensure that the water temperature and flow rate are at the desired levels. If flow rate has decreased it will have a major impact on the cooling potential, especially if the flow is no longer turbulent. Use a flow meter to evaluate the actual gallons per minute (GPM), but if a flow mater is not available disconnect the water return line and allow it to run into a bucket while timing it. Using the bucket method can give an idea of the GPM of flow, but remember that the water is free-flowing and does not have any of the closed-circuit back pressure that it can experience when everything is connected. Sometimes the poppet valve on the thermolator will not open to allow fresh cooling water into the circuit, which will cause the water temperature to rise over time.

Look for water lines that are not connected properly, for example a circuit that is looped without a water-in line. Check for any ball valves that are closed and blocking water flow. Beware of “vapor locks” or water circuits that have air trapped in them. All lines should be fully purged with water prior to starting up the mold. Verify water flow through all circuits.

A thermal image of a part after ejection can highlight areas of the mold that are running hot. If these areas do not match the original thermal images taken during process development there is likely a problem with the mold.

Another factor that must be considered is the temperature of plant cooling water. If running with a cooling tower the cooling water low temperature will vary over the seasons. In climates that have cold winters the cooling water may be 65 °F in the winter but only get down to 85 °F in the summer. If a molding process is established in the winter and the mold temperature is set at the minimum winter temperature the process will not be repeatable in the summer when the water is hotter. Set a minimum water temperature that is achievable all year. If the plant has a central chiller than the lower temperature can be achieved year-round.

Stay aware when a thermolator is swapped out for replacement. The replacement thermolator must have the same capability for flow and cooling capacity. If the machine was running with a 7.5-hp thermolator and it is replaced with a 3-hp thermolator the GPM capacity will be lower and there may not be enough flow to achieve turbulent flow. If turbulent flow is not achieved the cooling capacity of the water is greatly reduced.

41.3.1.7 Molding Process: High or Low Melt Temperature

Melt temperature is critical to a consistent process. A major part of the injection molding process is managing heat transfer. Understanding melt temperature and keeping it consistent and repeatable during molding runs will help minimize part variation.

Melt temperature should be documented on the setup sheet and verified during troubleshooting. If the melt temperature does not match the setup sheet (within ±10 °F due to measurement variation) investigate the settings that impact melt temperature including:

- Barrel temperatures
- Back pressure
- Screw recovery speed

If melt temperature is high the plastic will take longer to cool and will experience additional shrink that can lead to sink. The melt temperature impact can be especially important with semi-crystalline materials because if the material is cooled more slowly, larger crystals will be formed leading to higher shrink.

Low melt temperature can lead to an increase in pressure drop across the cavity and create a situation where pressure is not as well distributed across the cavity. Pressurization of the cavity is key to minimizing sink on the molded part. A severely low melt temperature will lead to a short shot.

41.3.2 Sink Troubleshooting Mold Issues

The root cause for sink can be the mold itself. Some of these factors include:

- Plugged cooling lines
- Poor cooling design
- Hot runner temperatures
- Valve gates not actuating
- Wall stock issues
- Cavity balance
- Gate size, location, and quantity
- Pull sink

41.3.2.1 Mold: Plugged Cooling Lines

To have effective cooling all water lines in the mold must be flowing well. If a water line becomes plugged there will be either restricted flow or no flow. This restriction in flow will lead to increased mold temperatures, which in turn could lead to sink.

Water flow through the cooling circuits should be measured and documented during process development. Once the baseline of cooling capability is established during process development it can always be referenced to when troubleshooting. If water flow through a circuit has decreased the cooling line may need to be cleaned out.

There are many things that can plug a cooling line including:

- Teflon thread tape
- Scale buildup
- Baffle inserted too far
- Metal shavings from machining process

Scale buildup in cooling lines can be another major cause of hot spots in molds. If the plant has water that is not treated properly mineral deposits will form inside the cooling lines. Scale buildup in a cooling line is an insulator and will restrict the cooling ability of the water traveling through the mold.

A molding plant must have a quality water treatment plan in place. If the plant runs without water treatment, scale buildup will be the result (see Figure 41.4). At its worst, scale buildup will choke off water circuits and severely limit cooling. Also understand that if the molds are experiencing scale buildup like this so is the heat exchanger and feed throat of the molding machine. Overheating oil may be the next issue that scale leads to.

If a mold has scale buildup, the mineral deposits must be removed either mechanically by drilling or chemically with an acid flush.

Figure 41.4 Severe water line buildup

41.3.2.2 Mold: Poor Cooling Design

If the mold does not provide adequate cooling for thick areas or part details the mold will run hotter in these areas. Hot spots in the mold tend to lead to sink problems. Whenever possible use baffles, bubblers, or heat pipes to provide enhanced cooling. Use of alternate materials such as copper-based alloys can also improve cooling in locations as long as the alloy insert can be reached with cooling water.

41.3.2.3 Mold: Hot Runner Temperatures

If hot runner temperatures are either set or running higher than desired the plastic in the manifold can pick up extra heat that can lead to sinks.

Verify that all hot runner zones are set to the documented temperatures and reading accordingly. Be watchful of zones that are calling for current 100% of the time or not at all because this most likely indicates that something is wrong with a zone in the hot runner system.

41.3.2.4 Mold: Valve Gates Not Actuating

Using valve gates allows molds to mechanically seal the gate with the valve pin. If the valve pin is not actuating and moving to its forward position the gate will not be sealed. Because the gate is open the plastic in the mold can flow back out of the cavity resulting in sink.

Verify that all valve gates are connected properly. Check full forward position in mold to verify that the valves are cycling fully forward. A visual inspection of the pins in the forward or closed position will show if the pins are moving correctly.

If all connections are correct and the valve is not cycling try to manually cycle the solenoid valve to see if this will make the pin move. If the pin moves when the solenoid is manually triggered inspect the open/close triggers or timers for the gates in question. If the pin does not move when the solenoid is manually triggered it may indicate that the air or hydraulics are not plugged in, the cylinder may be leaking, or the solenoid valve may not be shifting.

41.3.2.5 Mold: Wall Stock Issues

Maintaining nominal wall stock is one of the fundamental design requirements for plastics. Assuming that the part was designed correctly there can still be thick areas of wall stock in the mold. Any thick area will experience more shrink than the surrounding wall stock, and the thick areas often cannot be adequately packed because thin wall stock between the gate and thick areas will freeze off before the thick area is packed. Examine wall stock either with a micrometer or by cutting a cross section. Gates should be located in thicker wall stock regions to avoid gating thin to thick and experiencing freeze off prior to packing the thick areas. Use caution when gating into details such as ribs that are thinner than wall stock because they will freeze before the nominal wall is packed out, and only opening the gate will not improve this problem.

There are a few things that can lead to mold-related wall stock concerns such as:

1. Core pins that have been cut too short leaving a thick section

   If the sink is occurring behind a core pin check the depth of the core pin on the molded part by either using a set of calipers to check depth or cut through the part in the area of concern to check for thick spots. If the pin is short it must be replaced to eliminate the thick section.

2. Adding excess radius on the base of a rib

   If the rib to wall stock ratio was close to the maximum allowed for the material and extra radius is added the rib may sink. This is a difficult problem to correct because it requires welding in the mold. Use what is called the ball test and imagine a ball that is rolling through the nominal wall; if the ball hits an area where it drops and would hang up you can expect to see sink in this location. Figure 41.5 shows a pictorial example of the ball test. The circle represents a ball rolling through the nominal wall stock: a likelihood of sink is indicated if the ball stops rolling after it has hit the rib and dropped down. When paper cross sections are available a circle template is handy to conduct the ball test for wall sections.

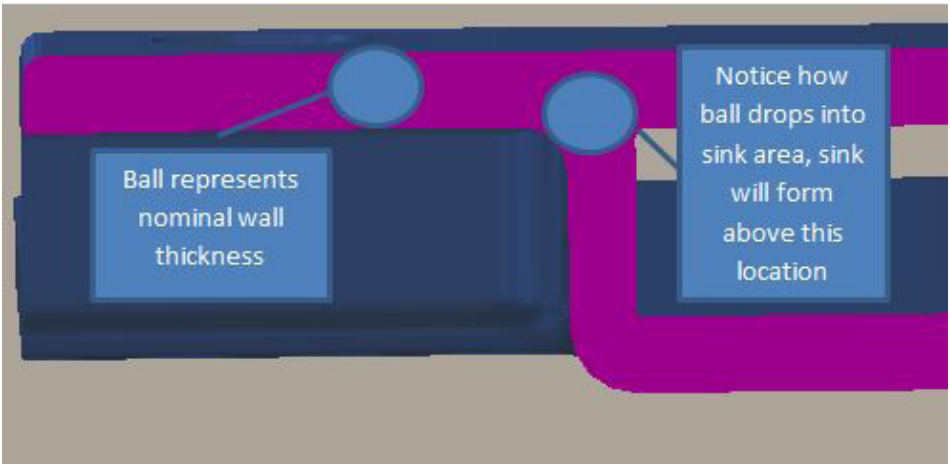


Figure 41.5 Ball test on wall thickness

41.3.2.6 Mold: Cavity Balance

Cavity balance problems can lead to situations where some cavities are packed out and others have sink. If a multicavity mold is displaying varying levels of sink by cavity the cavity balance should be examined.

See Chapter 12 on cavity balance for more information.

41.3.2.7 Mold: Gate Size, Location, and Quantity

If gate size is inappropriately small the gate may freeze before the part wall is packed out. There are many rules of thumb for gate sizing but they are rough estimates. There is a shear impact as material travels through the gate and the smaller the gate the more shear occurs; this can sometimes lead to a better packed part with a smaller gate.

Do not jump to the conclusion that the gate should be increased in size. There will be times that increasing the gate size only leads to increased cycle time without improving the sink on the part. Especially avoid changing gate size on a multicavity mold because this will create imbalances in shear rate, pack pressure, and gate seal time. Find the root cause rather than trying to balance cavity fill time by adjusting gate sizes.

If the gate is located too far from a part detail it will be impossible to pack the detail out. Verify flow lengths and distance from gates to make sure that end of cavity details can be packed out. In some cases it may be necessary to add additional gates to provide adequate pack pressure to all regions of a part. Also note that there will be cases where a rib or boss located close to a gate will actually freeze while the mold is filling and lead to problems with sink.

41.3.2.8 Mold: Pull Sink

Sometimes during ejection, a runner and gate will pull against the part and actual create a sink at the gate. This is more typical with sub gates or cashew gates.

If the gate is pulling a sink during ejection first verify gate seal. If the gate is sealed it is possible that the part is being ejected before it has cooled enough. Try adding cooling time to see if the defect is improved. If cooling time improves the sink look for ways to improve the cooling in the gate area and runner to shorten the cycle. This is another area that IR thermal imaging is useful.

Another possibility is that the gate is too thick to separate well during ejection. A thick sub gate or cashew gate will tend to not break as easily which can lead to a pull sink. If the gate is too thick it should be welded up and recut. Often thinner and wider gates will provide just as much volume throughput but freeze faster and break better.

Also if details such as bosses are sticking in the mold they may pull a sink opposite to their location during ejection. If the sink is directly opposite a boss check to ensure that the boss is polished correctly and does not have residual machining or electrical discharge machining (EDM) marks.

41.3.3 Sink Troubleshooting Machine Issues

Possible machine-related causes include:

- Machine performance
- Velocity to pressure transfer
- Inconsistent non-return valve
- Nozzle tip size/type
- Gas assist not working

41.3.3.1 Machine: Machine Performance

See Chapter 8 on machine performance.

41.3.3.2 Machine: Velocity to Pressure Transfer

Velocity to pressure transfer is when the machine transfers from first-stage velocity control to second-stage pressure control. If the machine struggles to achieve the set second-stage pressure after transfer the part may have sink. Figure 41.6 shows process monitoring curves for a machine running with poor velocity to pressure response. Notice how the machine pressure curve drops down in pressure after transfer and then takes several seconds to reach the set pressure. The cavity pressure curve drops in proportion to the machine pressure. In this particular case the

hold volume setting on the machine was set too low and the machine transferred as shown.

Figure 41.6 eDART® process monitoring showing a machine running with poor velocity to pressure response

Figure 41.7 shows poor velocity to pressure response on a machine controller. As is evident in Figure 41.6 the machine pressure drops well under the set point and then jumps above set pressure before finally stabilizing to the set pressure.

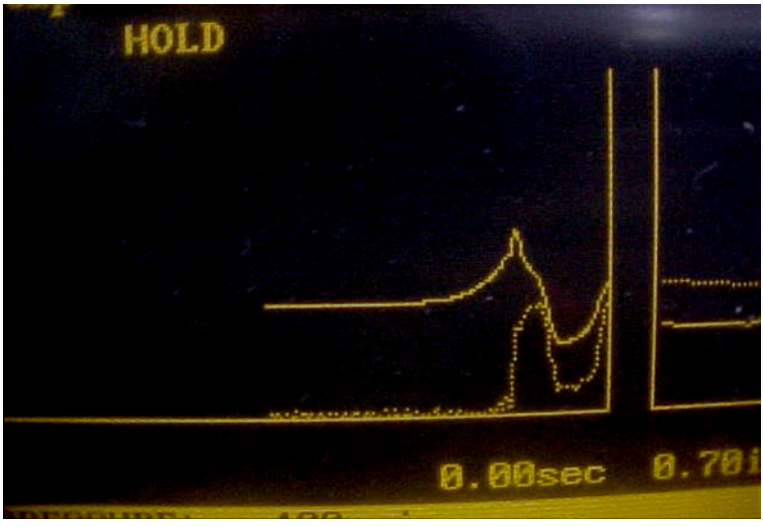


Figure 41.7 Velocity to pressure response on machine controller

Machine velocity to pressure transfer problems can come from a number of things including:

- Pack or hold volume, speed, velocity set wrong. If these parameters are set wrong the machine will not react well as it transfers. Adjust as required to provide a better transfer.
- Fill only weight too low. If the part is filled too short (below 95–98%) there will be a lack of resistance as the machine transfers, which will often result in a poor transfer. Adjust the fill only weight to match the documented process.
- Machine valve tuning set wrong. If the valve tuning is not set correctly the machine will not react correctly at transfer. Machine valve tuning may need to be adjusted to correct the problem.

41.3.3.3 Machine: Inconsistent Non-Return Valve

When a non-return valve is allowing plastic to leak, the cavity pressure will not reach the desired level. Because the cavity pressure is low the result can be sink. See Chapter 8 for details on measuring performance of a non-return valve.

If the non-return valve is leaking it will need to be replaced. There is a possibility that the non-return valve is worn, broken, or contaminated with foreign material.

41.3.3.4 Machine: Nozzle Tip Type/Style

If the machine has the wrong nozzle tip, style, or orifice the result can be an increased pressure drop that can lead to sink. The difference in actual cavity pressure can be substantial if the wrong nozzle or tip is used See Figure 41.8 for the cavity pressure difference between a 5/32-in and a 3/16-in orifice nozzle tip.

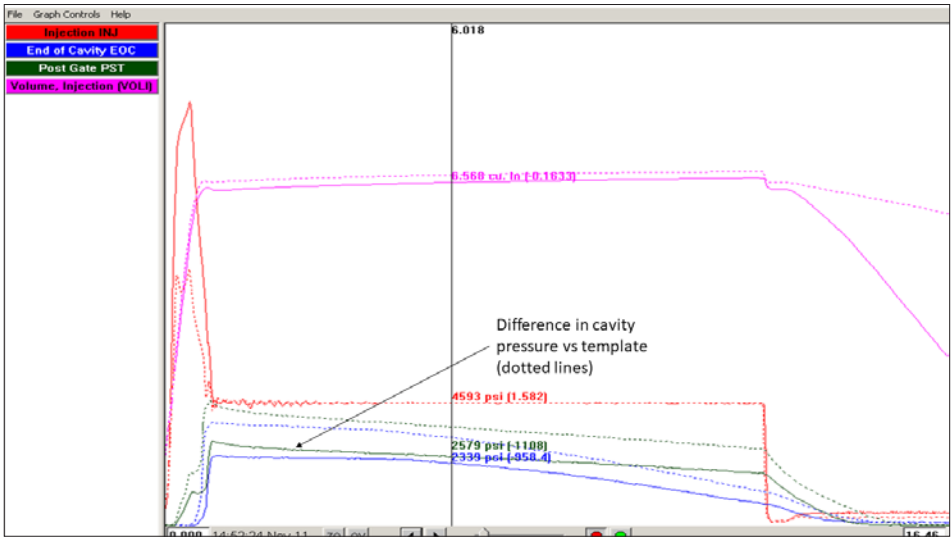


Figure 41.8 RJG eDART® cavity pressure data difference between a 5/32-in and a 3/16-in orifice nozzle tip

Part of the process setup must be verifying nozzle style and tip type/size. If the nozzle or tip is wrong it needs to be replaced before further troubleshooting efforts. The pressure drop of a mixing nozzle will be very different to that of a general-purpose nozzle. Do not assume that the nozzle and tip are correct when troubleshooting.

A simple trick that can help ensure that the correct tip is used is to tack-weld a 7/8-in nut on the side of the mold and then thread the required tip onto the nut. Every time a mold is set in a machine the tip is swapped. Anyone can walk up to the mold and with a quick look determine if the nozzle tip has been removed from the mold and presumably installed on the press.

41.3.3.5 Machine: Gas Assist Not Working

The full details of gas assist injection molding are beyond the scope of this book; however, this technique must be considered relative to sink defects. If the part is molded with gas assist and is displaying sink problems start troubleshooting the gas assist side of the process first. Some factors to check include:

- Fill only weight
- Gas inject pressure
- Gas pin function
- Overflow function
- Nitrogen level supplied
- Gas booster working
- Gas timing

41.3.4 Sink Troubleshooting Material Issues

Some potential material-related causes of sink include:

- Viscosity shift
- Nucleating agent
- Blowing agent

41.3.4.1 Material: Viscosity Shift

If the viscosity of the material varies enough it will impact the pressure drop across the cavity. Viscosity can vary due to variation in the incoming material or due to variation in moisture content (on a hygroscopic material).

A significant change in viscosity normally will also result in a change in transfer pressure. If the transfer pressure is running high compared to normal, the viscosity of the material may have increased. Verify the material incoming certification and check moisture content.

41.3.4.2 Material: Nucleating Agent

If the material being run is a nucleated semi-crystalline material the level of nucleation will impact the crystallinity of the material, which in turn can impact the shrink rate of the material. If the shrink rate is increased the result could be sink on the part surface.

Verify that the correct material is being molded. If using a nucleating agent masterbatch additive, check that the letdown ratio is correct.

41.3.4.3 Material: Blowing Agent

If a blowing agent is used to help pack out a part it must be added at a consistent level. If the blowing agent is not added at a consistent percentage the ability of the material to foam will vary. Verify that the blowing agent is added at the correct letdown ratio.

Also when using a blowing agent make sure not to pressurize the plastic with second-stage pressure. Over-pressurizing the material in the cavity will restrict the ability of the blowing agent to foam in the material. In some cases people will add second-stage pressure to the process, which can lead to additional sink because the function of the blowing agent is restricted.

42 Splay

42.1 Description

Splay is a defect where silver or white streaks appear on the surface of the part. Many defects get called splay. Therefore, an in-depth evaluation of the defect must be used to determine if you are truly dealing with splay. For an example of splay see Figure 42.1.

Also known as: silver streaks

Mistaken identities: scratches, scuffs, flow lines, or flaking/cold slugs

Figure 42.1 Splay Example

42.2 Splay Troubleshooting Chart

Table 42.1 shows the splay troubleshooting chart.

Table 42.1 Splay Troubleshooting Chart

Molding Process	Mold	Machine	Material
drying	venting	screw design	contamination
high melt temperature	hot runner temperatures	temperature control	excessively wet
decompression	cold slug wells	damaged components	wrong material
back pressure	sprue and nozzle tip orifice	cracked feed throat	blowing agent
screw recovery rate	sharp corners	lack of process control	
residence time	gate flaking		
screw recovery	lubricants		
feed throat temperature	cracked mold		
inconsistent material feed	leaking air		
	Venturi effect		

42.3 Splay Troubleshooting

The first consideration when trying to resolve a potential splay issue is to verify that the defect is actually splay. Many other defects get lumped into the category of splay. Below are several defects that are commonly misidentified as splay:

- Surface scratch from part handling
- Flow lines
- Delamination from contamination

In all troubleshooting efforts make sure that you witness the defect to verify the true problem!

Once you know that you are dealing with a true splay defect the first question to be asked is: “Is the splay located all over the part or just in a specific area?” The answer to this question will give the initial direction for your troubleshooting efforts.

- **Splay all over the part or at random areas usually means focus on the material or process side.**
- **Splay at specific and consistent locations usually means focus on the mold.**

Remember this is a general guideline for a starting point when dealing with splay.

42.3.1 Splay Troubleshooting Molding Process Related Issues

The following sections will deal with potential splay causes in the process including:

- Drying
- High melt temperature
- Decompression
- Back pressure
- Residence time
- Screw recovery
- Feed throat temperature
- Inconsistent material feed

42.3.1.1 Molding Process: Drying

One of the most common causes for splay is excess moisture in the material. This moisture will turn to gas when the material is subjected to molding temperatures. Figure 42.2 shows a magnified picture of splay on a part surface. This gas will be contained in the plastic melt until the plastic enters the mold, where the depressurization on the melt stream will allow the gas bubbles to reach the surface of the mold leaving behind the streak of splay. Hygroscopic materials, such as ABS, polycarbonate, nylon, TPU, polyesters, or PC/ABS, absorb moisture from the environment and require drying.

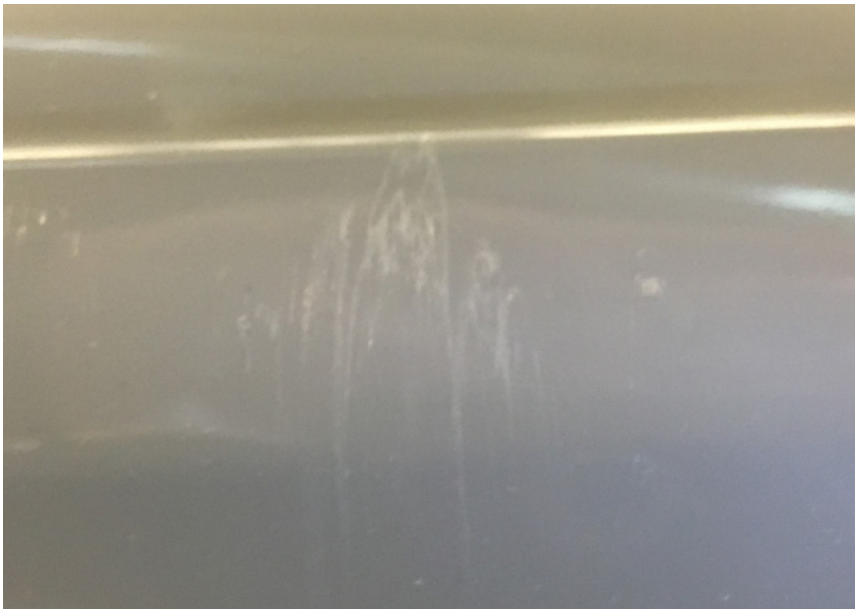


Figure 42.2 Splay magnified

Some additives such as talc can result in non-hygroscopic materials needing to be dried.

Material suppliers will provide a recommended drying temperature and time for hygroscopic materials. It is critical to follow these drying specifications to ensure that the material is dry enough to successfully process.

Successful drying requires the following:

- Temperature
- Dry air
- Air flow
- Time for the material to dry under the drying conditions

To ensure adequate drying it is critical to have all four of these conditions met. Four hours of drying time is meaningless if the temperature requirement is not met. A typical desiccant dryer will provide an air dew point of −40 °F. Keep in mind that the dryer temperature will help release the moisture from the pellets, the low dew point will allow the air to pick up the moisture, the air flow exposes more of the pellets to the warm/dry air, and the time is required to give the moisture a chance to leave the pellets and provide an adequate moisture content. For further details on drying see Chapter 9.

Material moisture content can be verified prior to molding by utilizing a moisture analyzer. The common types of moisture analyzers utilize two different techniques:

1. Carl Fischer titration method

   This technique relies on precise analytical equipment and requires chemicals to run. This test will provide a true moisture reading from a plastic.

2. Loss on weight method

   This technique uses a very precise scale to weigh the material at the start of the test. The material will be heated forcing it to give up moisture and the moisture analyzer will calculate moisture percentage based on this weight loss. Note that this method is more convenient but is less accurate because it will also measure other things that leave the material including residual monomers or low molecular weight additives.

Every plant must evaluate their choice of moisture analyzer. Carl Fischer titration will provide more accurate results but the cost of that is measured by the fact that it is a lab-style piece of equipment that takes more skill to run and maintain.

It is important to understand that just because a material has been dried to an adequate level does not mean it will stay there. If you expose dried material to ambient air that has humidity in it the material will begin to absorb the moisture like a sponge. Many materials will be too wet to effectively process if exposed to ambient humidity for longer than 15 minutes. To avoid this creating splay problems,

ensure that minimal material is allowed to be outside the drying hopper as it is transported to the press. This can be accomplished in a couple of different ways including:

- Use of small one-shot hoppers on the molding machine. This will keep the volume of material outside of the drying hopper minimized. Set the proximity switch on the one-shot hopper according to the required shot volume.
- Place the drying hopper on the feed throat of the machine. This will eliminate any issues with external transfer of the material from the drying hopper to the machine hopper. This works reasonably well with small hoppers but as they get larger cleaning them out during material changes can be more challenging.

Case Study: Drying

TPU is a very moisture-sensitive material and must be under 0.02% moisture content to process correctly. TPU will undergo a hydrolysis reaction that will lead to molecular weight reduction. When this degradation occurs the viscosity of the material will drop significantly, which will show in the transfer pressure on the machine. TPU also becomes extremely sticky when it is run wet and will be very difficult to remove from the mold. Past experience with TPU using both Karl Fischer titration and loss on weight moisture analysis showed that during humid months 5 minutes was the maximum time outside the dryer before the material was above the 0.02% specification! Make sure that everyone understands how critical it is to keep the material dry.

42.3.1.2 Molding Process: High Melt Temperature

Thermoplastic materials have a given temperature range within which they can be processed. If a material is processed outside of its specified temperature range thermal degradation can occur. When a material degrades it will create an off gas that can be trapped in the melt stream resulting in splay on the surface of the part. Figure 42.3 shows a degraded piece of material that created a splay streak down flow.

One of the first things to check relative to a temperature issue is the barrel set points. Verify that the barrel set points are set correctly (match setup sheet). Secondly verify that the actual thermocouple readings are within 5 °F of the set point. If a zone has a reading outside of the norm it should be checked to determine if the heater band is working. Often a barrel temperature will be running 20–30 °F above the set point, which could indicate that there is high shear in the barrel creating a temperature override. This high shear may be coming from excessive back pressure, high screw recovery rates, or improper barrel temperature profiles. When a heater zone is overriding the set point the process is running out of control.

Figure 42.3 Microscopic picture of slug of degraded material that created a splay streak behind it

When troubleshooting a process for splay it is critical to verify that the plastic melt temperature is within the recommended specification. Keep in mind that both too high a temperature as well as too low can lead to splay issues although too hot is the most typical culprit. Always keep in mind that temperature settings on the machine may not reflect the actual melt temperature of the material. There are cases where the melt temperature can be 50 °F or more over the barrel set points all due to shear heating from the screw.

There will be times when a heater band is burned out. You can detect these burned out heaters by checking the current draw using a clamp-style ammeter. If a heater band is burned out the barrel will have a cool spot and will not be uniformly heated. Uniform heating with consistency from run to run is critical in maintaining a repeatable, scrap-free process.

42.3.1.3 Molding Process: Decompression

Sometimes running with too much decompression (suck back) on the screw can cause ambient air to be drawn into the nozzle orifice of the machine. This ambient air will then often be deposited on the surface of the molded part as splay. If a large decompression is required to avoid drool from the nozzle tip it may be necessary to work through why so much decompression is required, and often times the cause is either too much heat or degraded material.

42.3.1.4 Molding Process: Back Pressure

Back pressure is an important process setting to generate a high-quality melt. A lack of back pressure will result in an inhomogeneous melt that may contain air. Part of the melt compression that takes place during melting forces the air that is trapped between unmelted pellets to be compressed and forced out of the melt. Back pressure is typically set from 500 to 3000 psi depending on how much shear and mixing is desired for a given process. When running below and even sometimes at the low end of this range splay can be the result.

One thing to watch carefully is when people reduce back pressure while purging a machine. This is done to help the screw recover but it must be changed back to the set point when the machine is running parts. This can often be overlooked, and an alternative to lowering the back pressure is to purge and use the manual decompression button to retract the screw to the desired shot size. If you change the back pressure there will come a time when you forget to reset it to the desired set point.

When back pressure is set too high it is possible to overheat the material, which in turn degrades the material. As mentioned in Section 42.3.1.2 melt temperature is important and the shear created from back pressure is a critical component of the melt process. High back pressure settings are often used as an aid to enhance color mixing to compensate for an inadequate screw design, and the result of this processing around items is a narrowed process window in other areas.

42.3.1.5 Molding Process: Residence Time

Residence time is defined as the time the plastic spends heated in the barrel of the molding machine and is determined by two parameters:

1. Shot size of mold versus barrel capacity of molding machine
2. Cycle time

As both of the above numbers increase so does the residence time. Many materials will degrade when held at melt temperatures for too long a time. The hotter the melt temperature the less tolerance there is for excess residence time.

Ideally the shot size for a given mold should be between 25–75% of the rated barrel capacity depending on the material being molded. When residence time goes up the risk of degrading the material rises. The best option when this situation occurs is to try to keep the melt temperature on the lower end of the recommended range, and this is done through shear input and barrel temperature set points.

42.3.1.6 Molding Process: Screw Recovery

When the screw is recovering there is a large amount of energy input to the plastic from the shear between it and the barrel. As the screw recovery rate increases the amount of shear also increases, which can drive the melt temperature of the mate-

rial past its upper limit. The screw should recover approximately 2–3 seconds prior to clamp open. An analogy of running the screw fast is using a kitchen mixer to whip air into a meringue.

42.3.1.7 Molding Process: Feed Throat Temperature

If the cooling water circulating through the feed throat is running too cold (below the ambient dew point) it is possible to have condensation of water in the feed throat. As this condensation drips into the feed throat the water gets conveyed down the barrel with the plastic. Feed throat temperatures should be controlled between 120–150 °F to firstly avoid condensation and secondly to avoid too much heat causing pellets to stick together and jam the feed throat.

42.3.1.8 Molding Process: Inconsistent Material Feed

If the loading of raw material is inconsistent the rotating screw can be starved of material as it tries to recover. This inconsistent material feed can lead to excess recovery time, which can lead to overheating of the material and in turn to splay. Always try to keep the hopper loading consistently. Inconsistent hopper loading can be detected in one of two ways:

1. If the loader keeps running cycle after cycle, it is usually a sign that the loader is having an issue keeping up with the machine. The root cause of this may be a low supply of material, plugged loader filter, or a feed wand that is not reaching the material.
2. Inconsistent screw recovery time.

42.3.2 Splay Troubleshooting Mold Issues

If the splay is in a consistent location the root cause will likely be a tooling issue. The critical step in troubleshooting tooling-related splay is too look upstream in the material flow to see what may be causing the splay. Often times the splay will appear on the part surface several inches away from the mold detail that is the problem. There are many factors involving the mold itself that can contribute to splay including:

- Venting
- Hot runner temperatures
- Cold slug wells
- Sprue and nozzle tip orifice
- Sharp corners
- Gate flaking

- Lubricants
- Cracked mold
- Leaking air
- Venturi effect

42.3.2.1 Mold: Venting

One of the biggest concerns with tooling issues creating splay is the impact of venting. Often times by allowing the gas to escape the mold through vents splay will be eliminated. If wiping the parting lines of the mold seems to improve the problem for a short period of time look at optimizing venting. All gas that is generated in the plastic melt stream as well as the air in the cavity must be allowed to exit the mold through vents. Many molders underestimate the importance of proper venting (see Chapter 7 on venting methods and design).

Case Study: Splay Caused by Lack of Venting

The splay was near the start of fill approximately 2–3 in away from the gate and to one side. The splay did not appear to be coming directly from the gate. The tool went back and forth to the tool room multiple times to review the venting; it was eventually vented well beyond the original venting, to the point where the whole perimeter of the parting line was now being vented. Feedback from processing was that the issue went away when aluminum tape was added on the parting line near the gate leading them to believe there was still an issue with lack of venting. Closer observation of where the aluminum tape was being placed, which was near the runner, showed this area was not vented. The mold was pulled from the press and vents were added to the runner. When the tool was set in the press and restarted the splay was completely gone! Venting on runners is as important as it is on the part (see Chapter 7 on venting methods and design).

Case Study: Splay from Screw Boss

In this case the screw boss was 1 in deep and with a shutoff hole in the center. The splay was occurring downstream from the boss. The mold was pulled from the press so the tool room could investigate venting options. The boss had an ejector pin and a sleeve that were not vented. The tool room added a ring vent to both the pin and the sleeve. When the mold was run again the splay was gone. If the boss has core detail cut in the solid a small vent hole can be added in the core detail and vented to the shutoff (see Chapter 7 on venting).

42.3.2.2 Mold: Hot Runner Temperatures

Hot runners are really an extension of the machine barrel and as such they can cause overheating of the material. Just like a machine barrel can overheat the material so too can the hot runner system.

One key item with hot runners and splay issues is verifying that the hot runner temperatures are set correctly. If someone accidently enters the wrong value for a set point the material can degrade causing splay. Also verify that the zone is correctly reaching the set point and is not running under or over the setting.

Sometimes detailed review of the hot runner manifold is required to determine if thermocouple zones have been mislocated. In this case either during mold build or mold maintenance a thermocouple wire is located on the wrong terminal in the plug. If thermocouple wires are "crossed" the hot runner controller will be trying to feed amperage to zone "A" but not seeing the feedback from the thermocouple because the zone "A" thermocouple is actually reading zone "B". This will lead to potential cold or hot spots in the hot runner manifold. Newer hot runner controllers will detect many of these types of issues, but old controllers will not. One way of detecting this issue with old hot runner controllers is to manually turn on one zone at a time and watch if the thermocouple of that zone reacts and shows a heat rise. If it appears that there is an issue with thermocouples being miswired, the mold must be corrected. Also bear in mind that a hot runner controller cable can be miswired from one plug to the other, which will lead to the same problem.

If a hot runner is built with corners that have dead spots or mismatches in the flow channels material will be trapped in these locations which will lead to potential splay troubles. Hot runner channels should be as consistent as possible with no dead spots that will collect material. Unfortunately, this is a problem that must be addressed at the time the manifold is built, because once you have it in the mold and in the press there will not be much of a choice but to process around the root cause. Fixing this issue after the manifold has been built will be a very expensive proposition, so make sure to verify this up front before the manifold is built.

If a hot runner manifold develops a leak due to damage, the result will be another spot that will trap and degrade material. Very often hot runner leaks occur from poor startup procedures. It is important to allow a hot runner manifold to heat up uniformly prior to starting to inject plastic through the mold. If a hot runner is cold started it will raise the potential of causing a crack in the manifold. Also remember that hot runners are built to function in a specific temperature range and the fit of components is built to accommodate the thermal expansion at this temperature. If the hot runner is set at a different temperature the thermal expansion of the components may create gaps leading to splay.

Case Study: Hot Drop Tips

This was a polycarbonate part with four low-vestige hot drop tips, and the splay was appearing near the gates. The part needed to be filled fast to fill and pack out the part, but at these high velocities splay became a problem. Inspection of the gate and spreader tip showed that the gate orifice was 0.050 in and the spreader tip was 0.025 in, which resulted in a gate area of 0.00147 in^2. The gate orifices were opened up to a diameter of 0.0625 in, which resulted in a gate area of 0.00258 in^2 (a 75% increase in area). By opening this restriction at the hot drop tips the splay problem was resolved.

42.3.2.3 Mold: Cold Slug Wells

Cold material, flakes, and drool can contribute to defects appearing to be splay near the start of fill. Proper cold slug wells must be designed at the transitions from sprue to runner and gate. Runner pins should not have flash or the flakes could lead to defects that look like splay. Also verify that undercuts on sprue pullers are not too aggressive or they will leave behind flakes. Many of these issues with cold material, flakes, or drool can create cosmetic defects that can look like splay.

42.3.2.4 Mold: Sprue and Nozzle Tip Orifice

The industry has different standards for cold sprue and hot runner nozzle seat orifices. The most common sizes for nozzle tips are 1/8, 3/16, 1/4, 5/16, and 3/8 in, and standard cold sprue orifices will be 5/32, 7/32, 9/32 in, etc. The standard cold sprue is designed to accommodate the nozzle orifice with a 1/32 in of clearance, which helps prevent sprue sticking. So a mold with a 5/32-in sprue orifice will need a 1/8-in nozzle tip for optimized flow without sprue sticking.

On the other hand, hot runner orifices are designed to a nominal flow channel without the clearance, so the nozzle tip should match the hot runner orifice one to one. This allows the hot runner to function as an extension of the machine barrel and reduces dead spots in the flow path. If a dead spot exists between the tip and the hot runner orifice, material will tend to degrade, which will lead to splay. Sometimes the root cause of splay is that someone forgot to switch the nozzle tip and the mismatch is causing these dead spots. Nozzle orifice should be documented for every mold and the tip should be changed whenever necessary.

Never run a mold with a nozzle tip orifice larger than the sprue opening on the mold. This will create sticking issues on a cold sprue and create a shear point and dead spot on a hot runner orifice.

42.3.2.5 Mold: Sharp Corners

Another issue with tooling that will lead to splay is sharp corners. Sometimes as the material flows across a sharp area splay will appear downstream. This can frequently be improved by softening the sharp corner and adding some radius.

42.3.2.6 Mold: Gate Flaking

Improper sub gate or cashew gate design can contribute to flow lines or defects with a splay-like appearance; this is due to plastic flaking from the gates as they separate from the part. If this is the suspected root cause you may be able to detect flakes of plastic on the mold near the gate. A useful trick to help diagnose this problem is to manually blow air through the gate between cycles. See Chapter 4 on gate and runner design.

If flaking is the root cause magnification of the defect will show that the starting point of the splay has a visible speck.

42.3.2.7 Mold: Lubricants

Sometimes splay will come from lubricants used on the mold. There are cases where grease from lifters or ejector pins can bleed onto the core surface. This grease can be pulled onto the surface leaving behind a splay streak. Also lubricants and metal protectants can be trapped in inserts and bleed out onto the mold surface, which much like grease can be pulled onto the surface of the mold resulting in splay. If the splay shows up immediately after the mold has come back from the mold shop you may have an issue with too much lubricant resulting in splay. Avoid over-lubricating molds, because this is not a case where “if a little is good, a lot is better”. If you have a mold that is over-lubricated you may need to wipe the excess lubricant out, and may even have to disassemble the mold to solve the problem.

42.3.2.8 Mold: Cracked Mold

Another tooling issue that can cause splay is a crack in the mold. If a mold cracks to a water line the water can wick to the surface of the mold resulting in splay. Cracks in the mold can be obvious and with water drips showing on the mold surface. However, there will be cases where the water only shows up after the mold has been closed and clamped up, so when checking for suspected cracks always close the mold to see if the clamp pressure opens up the crack. A short-term fix for a cracked mold is to use a reverse-flow thermolator; but this will affect the cooling capability of the mold. The ultimate fix will be to repair the crack. Be aware during tool design that sharp radii will be more prone to crack.

42.3.2.9 Mold: Leaking Air

If the mold is relying on air poppets for ejection, splay may appear from the poppet leaking as the material is injected into the mold. Poppets should only be energized when the part is to be ejected from the mold. Poppets that are leaking air will introduce air into the melt stream potentially forming splay.

If running a gas-assisted part, a leaking gas pin can allow nitrogen to enter the melt stream resulting in splay. Also if the gas is turned on too early splay could be evidenced in the part. Running gas assist on parts creates another level of complexity and must be considered during the troubleshooting process.

42.3.2.10 Mold: Venturi Effect

There will be cases where the plastic flowing in the mold will pass over a rib rather than filling into the rib. The plastic may pull some of the air trapped in the rib up into the melt stream producing splay on the part. The key to avoiding this issue is to make sure that ribs are vented whenever possible. Fill velocity may also impact this issue but does not address the root cause and will narrow the process window.

Case Study: Splay from Ribs at End of Fill

This case was a polypropylene part with ribs approximately 2–3 in from the end of fill; there was a splay issue near the ribs that could not be eliminated. The mold was worked on to add radii to sharp corners at the ribs as well as adding ejector pinsin the ribs to help vent the gases out, but these changes did not impact the splay problem. To take all possible root causes off the list the decision was made to add perimeter venting to the whole end of the part. A vent track was added with a 0.060-in land and the vent was ground open to 0.001 in. When the mold was rerun the splay was gone; however, it came back a couple of days later. When the mold was reviewed again it was observed that the vents were already closed off. The mold had very little parting line bearing surface with too much relief, which allowed the parting line to hob in and close off the vents. When the vents were opened the splay again disappeared!

42.3.3 Machine Issues

The root cause of splay can sometimes come from issues with the machine itself. These issues tend to be rarer but can be the case where you try “everything” but cannot find a solution. Below are some potential machine-related causes to splay:

- Temperature control
- Screw design

- Cracked feed throat
- Damaged screw, barrel, or non-return valve
- Lack of process control

42.3.3.1 Machine: Temperature Control

Issues with heat control on the molding machine can be a frequent cause of splay on molded parts. Both too high and too low a material temperature can result in degradation of the material that may result in splay. When temperatures are too high thermal degradation of the material will occur, which will generate outgassing that leads to splay. When the temperatures are too low excess shear will occur in the screw and barrel, which can also cause material degradation. Most newer machines have heater band burnout detection, which will help with these potential issues. If you do not have heater band burnout detection on the machine you will need to check current draw on the heater bands with an "Amp Clamp" type of meter.

42.3.3.2 Machine: Cracked Feed Throat

A crack in the feed throat can allow feed throat cooling water to leak and get into the incoming stream of material. This rarely occurs and is difficult to diagnose because you will have to clear the feed throat and remove the hopper to see if there is a crack and water leaking. No matter how well you dry the material if you are adding water back in at the feed throat you will have splay problems.

42.3.3.3 Machine: Screw Design

Poor screw design can lead to poor melt quality, which in turn can result in splay. The primary example of this would be running a semi-crystalline material in a general-purpose (GP) screw with an inadequate compression ratio or L/D ratio. This combination will lead to poor melt quality where in the worst case the melt stream has unmelted pellets that can often leave a surface defect with a trail of splay. At a lesser level the air that is between the unmelted pellets will not be compressed out of the melt stream, which will result in gas being compressed into the melt stream.

Keep in mind that a GP screw is at best a compromise trying to be a "one size fits all" item. Depending on the material being run a specific compression ratio, L/D ratio, and mixing elements may be required to provide an adequate melt without gas entrapment. Figure 42.4 shows an example of a standard GP screw.

Figure 42.4 GP screw

42.3.3.4 Machine: Damage to Screw, Barrel, or Non-Return Valve

Damage to the screw, barrel, or non-return valve can all lead to hang up areas as well as high shear locations that can lead to splay. Any area that traps material can result in material becoming degraded; as the material degrades it can release gases that will be trapped in the plastic melt stream and cause splay. Whenever a screw is checked for preventative maintenance, a part of the procedure should be to look for any damage to the screw, barrel, and non-return valve.

Care must be taken when a screw is cleaned. There are cases where cleaning the screw with a grinder leaves a scratched and rough surface. Copper gauze is typically recommended for cleaning of a screw because the copper will not scratch the screw surface. If a screw becomes scratched or otherwise damaged there will be an increased tendency for material to hang up resulting in splay, brown streaks, or black specks.

42.3.4 Material Issues

Some splay problems are directly caused by the material itself. Below are potential causes based on material problems:

- Contamination
- Excessively wet
- Wrong material
- Blowing agent

42.3.4.1 Material: Contamination

Contaminated material is the most likely material-related problem encountered. The raw material can be contaminated with other materials, dirt, cardboard flakes, or other potential sources. Any source of foreign material may lead to splay. The

contaminant may degrade at a different temperature, which results in outgassing causing bubbles in the plastic melt stream. Also contamination by foreign materials may cause a surface that delaminates giving an appearance similar to splay. Material contamination can come from anywhere along the feed stream of the material including at the supplier, in the tanker truck, in the silo, in the gaylord, in the drying hopper, or at the machine hopper. Always take precautions to avoid cross contaminating material when loading it into a silo or hopper.

Figure 42.5 Open gaylord: potential for contamination

42.3.4.2 Material: Excessively Wet

Excessively wet material can lead to severe problems. Even non-hygroscopic materials can have splay problems when molded with a large enough amount of surface water. Reach your hand down into the material; if it is wet enough you will feel the moisture. Even though you normally would not dry this material (polypropylene for example) it will need to be dried to get rid of this surface moisture.

Case Study: Splay on Polypropylene

This was a case where splay started to show up on an unfilled polypropylene part that never previously had splay. During process troubleshooting it was determined that the gaylord of material had gotten stored under a leak in the roof. The material was literally dripping wet when it was loaded into the machine, which created the splay. This material was dried for a couple of hours, which eliminated the splay. There are even situations where silo material can become wet from a leak or an open hatch.

42.3.4.3 Material: Wrong Material

It is always worthwhile to ensure that the correct material has been loaded into the molding machine. Many defects can be created if the wrong material is added to the molding machine. Verify that the material labels match the required material on the bill of materials. Be aware that this can create hazardous conditions due to severe material degradation.

42.3.4.4 Material: Blowing Agent

When using a blowing agent or microcellular foam process, gas is of course introduced into the plastic in the barrel intentionally. This gas results in a foamed material that will often be used in thick cross sections to avoid sink and reduce weight and cycle time. Because gas is being added to the material splay will be evident across the whole part. There are things that will improve the surface of the part such as gas counter pressure but it will be very difficult to completely remove all of the surface splay when running with a blowing agent or microcellular foam.

If blowing agent is introduced into a material that was not intended to be foamed the result will be splay. Treat this like any other contamination in the material.

43 Sprue Sticking

43.1 Description

The sprue is the part of the runner system that provides entrance into the mold from the nozzle of the molding machine. A sprue is considered to stick when the mold opens and the sprue remains behind in the sprue bushing on the "A" side of the mold. See Figure 43.1 for an example of a typical sprue.

Figure 43.1 Typical sprue

43.2 Sprue Sticking Troubleshooting Chart

Table 43.1 shows the sprue sticking troubleshooting chart.

Table 43.1 Sprue Sticking Troubleshooting Chart

Molding Process	Mold	Machine	Material
high second-stage pressure	undercuts in sprue bushing	nozzle problems	moisture content
long second-stage time	incorrect finish in sprue bushing	machine performance	material type
short cooling time	sprue puller	injection unit forward pressure	additives
high mold temperature	cooling of sprue bushing		
low nozzle temperature	sharp corners		
sprue break	oversized sprue or runner		

43.3 Sprue Sticking Troubleshooting

Sprues stick for a wide variety of reasons. When a sprue sticks the next shot will not shoot, which in turn should cause the machine to alarm out. Removing a stuck sprue normally involves heating up a brass screw and sticking it in the end of the sprue, waiting for the plastic to solidify around the screw, and then using pliers or a slide hammer to pull on the brass screw hopefully popping the sprue out. When trying to remove a stuck sprue always check the sprue bushing radius to determine if there is any flash present. Flash around the end of the sprue should be removed prior to trying to pull the sprue (use a heated brass rod to melt away the flash).

43.3.1 Sprue Sticking Troubleshooting Molding Process Issues

Sprue sticking process problems can include:

- High second-stage pressure
- Long second-stage time
- Short cooling time
- High mold temperature

- Low nozzle temperature
- Sprue break

43.3.1.1 Molding Process: Second-Stage Pressure

If the second-stage pressure is high it can over-pack a sprue and cause it to stick. Because sprues are usually the thickest portion of the part and runner system a great deal of pressure can be packed into the sprue.

Verify that the second-stage pressure is set to the documented process. If high pressure is suspected running a fill only shot will validate the influence of pressurizing the sprue. If a fill only shot does not stick in the sprue it can point to issues with over-packing of the sprue that may be limiting the shrinkage that occurs in the sprue.

Low second-stage pressure can result in sprue sticking if the thick section of the sprue develops a void. Sprues with voids in the thick section will often tear as the mold opens. If the sprue is pulling apart and a void is present try increasing second-stage pressure to eliminate the void.

43.3.1.2 Molding Process: Long Second-Stage Time

If second-stage pressure is applied after the gate has frozen the only thing that is being pressurized is the runner and sprue. Excess second-stage time may lead to over-packed sprues that will stick. In addition to the potential for sticking sprues excess second-stage time leads to increased cycle times and heavier runners, both of which are costing the molder money. Furthermore, the machine will be expending additional energy to provide pressure for the extra amount of time, which will also increase cost.

Second-stage times should be evaluated versus the required time for the gate to seal (see Chapter 3 on scientific molding).

If the process is to run with gate seal it is a good idea to have an accommodation for variation that may be 5–10% added to the gate seal time to ensure gate seal occurs. Some companies will select a given amount of time to add to gate seal time to establish second-stage time. Choose a method and make sure that it is followed for setting second-stage time.

Examine the second-stage time and verify that it is set correctly. Also verify gate seal time and determine if the second-stage time is set appropriately. If there is extra time on second stage it will be worth reducing the time to determine the impact on the sticking problem. To keep the cycle time consistent the time removed from second-stage injection can be added to cooling time. As always, part requirements must be understood, and reduction of the cycle time for some customers may lead to revalidation of the molded parts.

43.3.1.3 Molding Process: Short Cooling Time

If the cooling time of the process is too short the sprue will not have enough time to adequately shrink and therefore may not release from the sprue bushing. Also if the cooling time is too short the sprue may be too hot for consistent extraction from the sprue bushing. Many times sprues that are too hot will actually separate from the sprue puller and stick in the sprue bushing.

Check that there is adequate cooling time to allow for proper removal of the sprue. Adding time to cooling can help validate if this is the cause of the sprue sticking. If adding cooling time helps the problem, examine the mold for opportunities to enhance the cooling in the region of the sprue.

43.3.1.4 Molding Process: High Mold Temperature

A hot mold will result in the same concerns as too-short cooling times. The sprue may not shrink enough for effective release, or the sprue may be too hot and actually tear apart as the mold opens.

Verify the mold surface temperature near the sprue bushing. If the mold is running hot determine if this is a new occurrence or if the mold has always run hot.

Check for cooling concerns including:

- Water temperature
- Water flow rate
- Water line setup
- Scale buildup in cooling channels

Any of the above cooling concerns can influence the cooling rate of the plastic and must be controlled to maintain consistent cooling for the life of the mold. If any of them has changed from the original process data the problem must be corrected.

Thermal imaging will help to detect hot spots on a molded part, runner, and sprue. A baseline thermal image should be taken during process development to help with troubleshooting in the future. Comparison of baseline images and current images may point out cooling problems with the mold.

43.3.1.5 Molding Process: Low Nozzle Temperature

Occasionally a low nozzle temperature will cause a large slug of frozen plastic in the tip that does not want to separate from the sprue when the mold opens. Raising the nozzle temperature can improve the break of the sprue to the nozzle tip.

43.3.1.6 Molding Process: Sprue Break

Machines have a setting called sprue break that allows the injection unit to move back prior to mold open. In some cases adding sprue break will allow the sprue to break cleanly away from the nozzle tip and may resolve a sprue sticking issue. If

using sprue break stay very aware of any nozzle drool as it will lead to damage of the tip or sprue bushing over time (and may well cause sticking).

43.3.2 Sprue Sticking Troubleshooting Mold Issues

Below are some of the potential mold concerns for sticking sprues:

- Undercuts in sprue bushing
- Incorrect finish in sprue bushing
- Sprue puller
- Cooling of sprue bushing
- Sharp corners
- Oversized sprue or runner

43.3.2.1 Mold: Undercuts in Sprue Bushing

Any undercuts in the sprue bushing may cause a sprue to stick. If the sprue bushing has been damaged from steel tools the undercuts will lead to problems. Sprue bushings are used reamed with a tapered reamer that will leave circular rings with the sprue bushing that can cause sticking. There will be cases where a sprue needs to be draw polished to allow proper release of the sprue. See Figure 43.2 for a magnified view of circular polish lines on a sprue that could lead to sticking; draw polishing the sprue bushing would remove these undercuts.

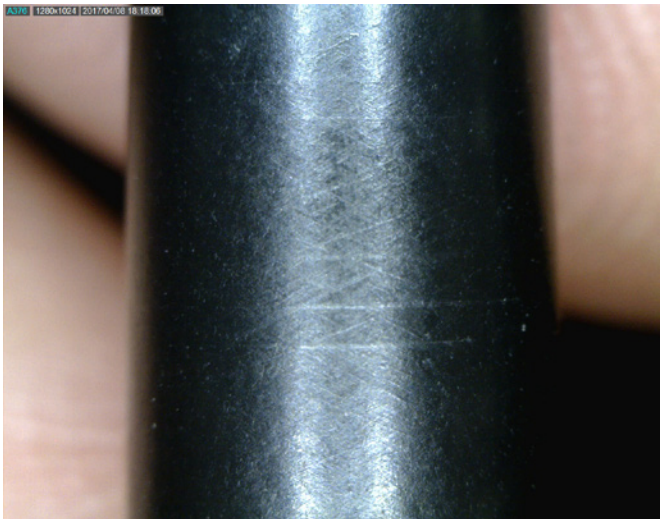


Figure 43.2 Magnified view of sprue: notice the circular cutter lines

Inspect the seat of the sprue bushing for damage. Plastic can leak between the sprue bushing and the nozzle tip if there is damage. The sprue bushing seat can be resurfaced if it has been damaged. Damage at the sprue bushing to nozzle tip will often show as flash at the end of the sprue.

Glass-filled materials may lead to erosion of the steel in the sprue bushing. This erosion creates an undercut that will cause sprues to stick. When dealing with erosion it is usually best to replace the sprue bushing rather than trying to polish away the erosion.

43.3.2.2 Mold: Incorrect Finish in Sprue Bushing

As with general sticking some materials will need a draw polish and some will need a blasted finish for the sprue bushing. The perfect example is a TPU which will release much better with a sprue bushing that has been sand blasted. If the sprue is sticking without an apparent reason investigate alternate finishes in the sprue bushing.

Circular polishing of a sprue bushing will create small undercuts that may cause a sprue to stick. Proper finish will normally eliminate this sticking problem.

Many materials including polycarbonates, ABS, and nylon will usually work best with a sprue bushing that has been draw polished. In the case of polypropylene often times a shiny highly polished finish can lead to sticking. Do not assume that finer polish will resolve a sticking issue. Draw polishing a tapered sprue can be difficult but is often the solution to a sticking sprue. A 320-grit paper draw polish will often resolve sprue sticking issues.

43.3.2.3 Mold: Sprue Puller

There are different styles of sprue pullers to aid in removal of the sprue from the sprue bushing. The most common types are “Z” pullers and back drafted cold slug wells. In general, the “Z” puller will provide a more robust pull of the sprue. Back drafted cold slug wells can only be back drafted so far or they will not release from the ejector side of the mold. “Z” pullers are cut into the cold slug well ejector pin and should have ample radius to eliminate breaking at sharp corners.

Figure 43.3 shows an example of a Z-style sprue puller. The ejector pin in the cold slug well is ground with an undercut to capture the sprue and allow it to pull from the sprue bushing when the mold opens.

Figure 43.3 Z-style sprue puller

Figure 43.4 shows a back drafted style sprue puller with an added undercut ground into the steel to help with sprue sticking. These extra undercuts can help with sprue sticking but also may leading to flaking problems.

Figure 43.4 Back drafted sprue puller with ground in undercut; note that the cold slug well is undersized for the sprue

Figure 43.5 shows an example of a sprue from a mold with a broken Z puller. This can occur if the Z undercut is cut too deep resulting in a thin steel condition. Broken Z pullers will result in stuck sprues.

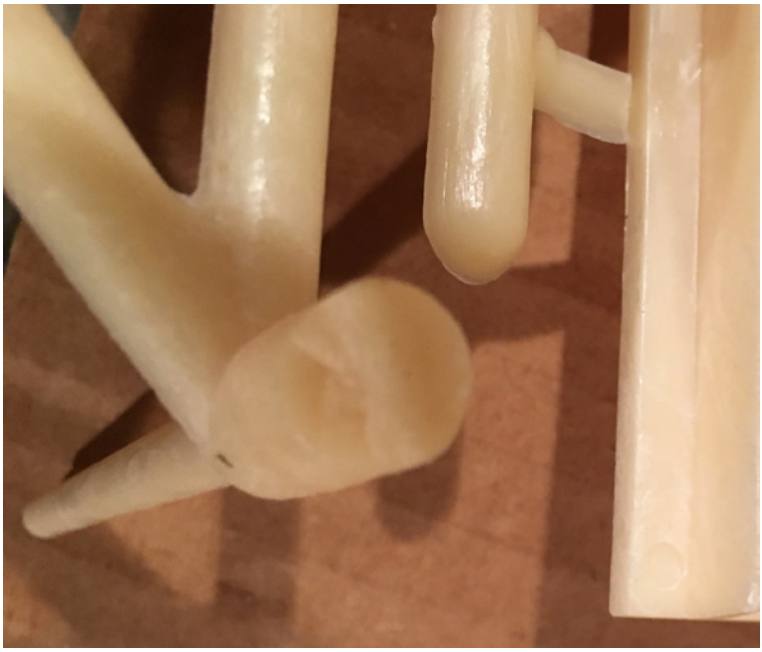


Figure 43.5 Sprue from a mold with a broken Z puller

43.3.2.4 Mold: Cooling of Sprue Bushing

Because the sprue base is typically the thickest area of plastic in any mold, cooling is critical for ensuring that the sprue is solid enough for removal. Cooling lines in the mold should be designed to provide uniform cooling around the sprue bushing. Also ensure that the sprue bushing has good contact with the cavity block to allow heat transfer from the sprue bushing to the mold. Any reduction in contact between the sprue bushing and the cavity block can lead to reduced cooling capability for the sprue.

High-performance sprue bushing made from copper alloys can improve the cooling of sprues if used correctly. Any time a high-conductive material such as copper alloy is used in a mold a cooling path must be provided to allow the heat to transfer out of the copper component. If cooling is not provided to high conductive inserts or sprue bushings the heat will transfer from the steel into the copper alloy but will have nowhere to go from there, the result of which is the alloy will heat up and make things worse.

Another consideration is to replace the sprue bushing with a hot sprue. The two big advantages of a hot sprue are:

1. Reduced cycle time due to not needing to wait for the sprue to cool.
2. Reduced waste in the runner system because the sprue stays molten instead of being cooled and ejected every shot.

43.3.2.5 Mold: Sharp Corners

All edges on the sprue and runner transitions should be broken to reduce stress points for breaking and reducing shear. This also includes sprue-style tips on hot runners. In cases where this does not help, thin gusset ribs can be added. These will cool quickly and add structure/support to the sprue runner transition. In cases where gusset ribs are needed, the root cause is usually excessive sized runner and sprue diameters. Adding larger radius to corners in these cases will add more mass and can make the problem worse.

43.3.2.6 Mold: Oversized Sprue or Runner

Sprue bushings and runners that are excessive in size create unwanted mass that will require excessive cycle time to cure to prevent the sprue from breaking off from the runner. This is very common in the industry and can contribute to unwanted waste with runner and sprue scrap.

Oversized sprues and runners will often lead to increased cycle time to allow for cooling. If the sprue and runner is overly soft at the desired cycle time it probably is oversized or does not have adequate cooling in the runner sprue area. There are many cases where the replacement of a sprue bushing with a smaller diameter sprue bushing would lead to dramatic drops in cycle time. The cycle time should be dictated by the part and not the sprue or runner cooling.

43.3.3 Sprue Sticking Troubleshooting Machine Issues

Potential machine-related causes of sprue sticking include:

- Nozzle problems
- Machine performance
- Injection unit forward pressure

43.3.3.1 Machine: Nozzle Problems

A damaged nozzle or nozzle tip can cause the sprue to stick. Some common problems are damage to the face of the nozzle tip which allows plastic to leak at the interface between the nozzle tip and the sprue bushing. Damaged nozzle tips can be resurfaced or replaced.

Another item to watch for is matching the nozzle tip orifice to the sprue bushing. The nozzle tip should be 1/32 in under the size of the sprue orifice to allow maximum flow through the tip and avoid the situation where the nozzle tip is oversized. An oversized nozzle tip orifice will result in sprues sticking. There are nozzle orifice gauges for checking tips and sprue bushings.

Also verify that the correct tip radius is being used. The common tip radii are 1/2 and 3/4 in. Make sure that the correct radius is on the nozzle tip. Most shops standardize on one radius but takeover molds may have something different. Also there are occasions where the sprue bushing radius is missing in a mold design review and the wrong radius shows up on a mold. There are radius gauges available to check the sprue bushing and the nozzle tip.

Inspect the injection unit for proper nozzle alignment to the mold sprue bushing. Place a piece of cardboard on the sprue bushing and bring the injection forward. Back off the injection unit and inspect the cardboard for uniform contact between the sprue bushing and the nozzle tip. Pressure-sensitive film can also be used in place of cardboard for a more accurate depiction of the contact force. The injection unit may need to be aligned if the nozzle is not hitting squarely. There are cases where the injection unit jumps as it locates into its full forward position, which indicates an alignment problem.

Figure 43.6 Nylon nozzle tip with undercut from erosion

When molding glass-filled materials the nozzle tip may have erosion problems that can actually lead to undercuts in the nozzle tip. This undercut area will in turn lead to sprue sticking. Examination of the sprue will normally highlight if there is an area that is undercut, and if an undercut is detected the nozzle tip should be replaced to eliminate the problem. Figure 43.6 shows a nylon tip with an undercut

that formed after running glass-filled nylon. The impact of glass-filled plastic on the steel can be hard to believe, but the high shear rates combined with the glass filler can create many issues if not accommodated.

43.3.3.2 Machine: Machine Performance

All set points on the machine are important to the process.

See Chapter 8 on machine performance.

43.3.3.3 Machine: Injection Unit Forward Pressure

If there seems to be leaking between the nozzle tip and the sprue bushing verify that the pressure for holding the carriage forward is set appropriately. This pressure may be called sled forward pressure, nozzle touch force, or nozzle forward pressure. If this pressure is set too low there may be leaking pressure between the nozzle tip and the sprue bushing which in turn can lead to sticking sprues.

43.3.4 Sprue Sticking Troubleshooting Material Issues

Some material concerns include:

- Moisture content
- Material type
- Additives

43.3.4.1 Material: Moisture Content

If processing with wet material the material can degrade and become very sticky. If processing a hygroscopic material verify that the material has been adequately dried. Materials that are very sensitive to hydrolysis such as TPU, polyesters, and polycarbonate tend to have more problems due to lack of drying.

See Chapter 9 for more information on drying.

43.3.4.2 Material: Material Type

One immediate thing to consider with material type is the wear caused by running glass-filled materials. The glass content can be very abrasive and will cause erosion damage to the steel in the sprue bushing. If this erosion becomes severe enough sticking sprues may be the result. If running glass-filled material investing in a hard sprue bushing may pay long-term benefits.

Some materials are inherently stickier than others. Materials such as nylon, polyesters, TPU, and polycarbonate can tend to stick harder in the sprue bushing. Flexible materials such as PVC or TPEs may be soft enough that they tend to tear or stretch and jump past the sprue puller resulting in a stuck sprue.

It may seem counterintuitive but when molding a material like TPU the sprue bushing should be bead blasted rather than polished to ensure release. If the sprue is sticking and the TPU material is known to be dry the sprue bushing may need to be blasted.

43.3.4.3 Material: Additives

A lack of release agent in a material will lead to sticking concerns. If sticking starts when a new lot of material is molded a call to the material supplier may be warranted. Do not get in the habit of "crying wolf" to the material supplier with every problem that pops up. Make sure that all internal root causes have been investigated and resolved. It is a major waste of everyone's time to call in a material technology representative for a splay issue with the material only to have them point out that the dryer is running with a +30 °F dew point.

If a release agent is added as a masterbatch at the machine check that it has been added and at the correct letdown ratio. Also evaluate if some colors stick worse than others, because this could point to different additives in the colorant package or a different nucleating effect depending on the color. Different nucleation in semi-crystalline materials can lead to different cooling and shrink rates and may impact how a sprue releases.

Glass-filled materials can have a tendency to cause erosion of the sprue bushing, which can create undercuts and sticking concerns. When running glass-filled materials use of a sprue bushing of higher hardness may be necessary to eliminate erosion concerns.

44 Stringers

44.1 Description

A stringer is a drawn-out filament of plastic that usually originates at a nozzle tip or a hot tip in a mold. Molten plastic will stretch out into a long filament when the mold opens. See Figure 44.1 for a typical example of a stringer on a sprue.

Also known as: strings

Mistaken identities: none

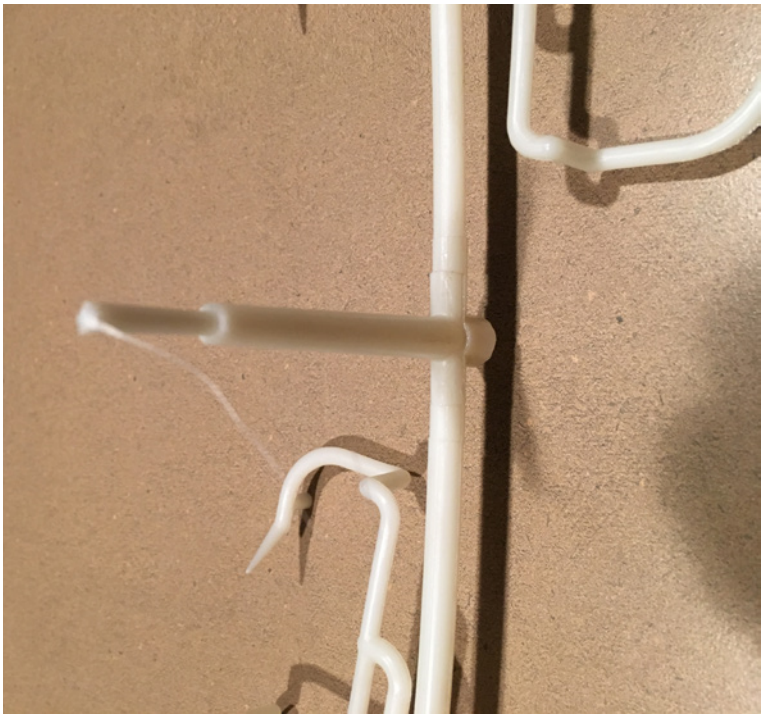


Figure 44.1 Typical stringer on sprue

44.2 Stringers Troubleshooting Chart

Table 44.1 shows the stringers troubleshooting chart.

Table 44.1 Stringers Troubleshooting Chart

Molding Process	Mold	Machine	Material
high nozzle temperature	hot tip temperature	nozzle heater setup	moisture content
low decompression	tip-bearing surface	machine performance	volatiles
high melt temperature	lack of cooling	nozzle tip	
	valve gates		
	tip orifice size		

44.3 Stringers Troubleshooting

Typically, stringers appear due to too much heat at a nozzle or hot tip. An important factor to consider with stringers is that they can lead to parting line damage over time. Stringers on the mold surface will actually create a hobbing of the steel resulting in mold repairs.

44.3.1 Stringers Troubleshooting Molding Process Issues

Potential processing-related concerns are:

- High nozzle temperature
- Low decompression
- High melt temperature

44.3.1.1 Molding Process: High Nozzle Temperature

If the nozzle temperature is set or running too high the material will remain molten at the nozzle tip, which will allow a stringer to be pulled as the mold opens. The plastic at the very tip of the nozzle must freeze off enough to create a break point for the material that will not be molten.

When experiencing stringers review the actual nozzle temperature against the specified set point. If the nozzle is running over the set temperature investigate why. Try reducing the nozzle temperature to see if the stringers are reduced. Figure 44.2 shows a thermal image of a nozzle and adapter.

Sometimes the only way to eliminate stringing is to add an insulator between the nozzle tip and the sprue bushing. This insulator is most frequently a piece of cardboard; however, there are molded insulator tips and Kevlar fiber pads that can also be used. The insulator will frequently solve the stringing problem.

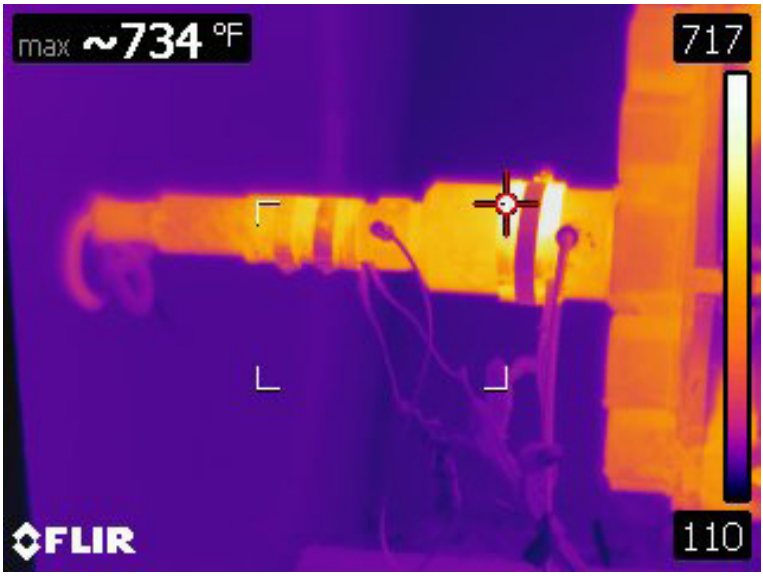


Figure 44.2 IR image of extended nozzle and adapter

44.3.1.2 Molding Process: Low Decompression

If the decompression setting on a machine is set too low the shot will have a higher residual pressure acting on it. This pressure builds during screw recovery and is relative to the amount of back pressure applied during recovery. This pressure on the melted shot will continue to act as a force against the plastic in the nozzle and nozzle tip. Excessive pressure on the shot will push the plastic at the front of the nozzle tip and may allow the material to string and also drool when the mold opens and the sprue leaves contact with the nozzle tip.

Increasing the decompression set point can help keep the plastic away from the end of the tip and in turn reduce stringing. Watch for excess decompression because it can cause splay problems.

44.3.1.3 Molding Process: High Melt Temperature

Even if the nozzle temperature is set appropriately the actual melt temperature of the material may be too high, which results in stringing. Melt temperature that is running high will make it more likely for stringing to occur.

Check the actual melt temperature of the material. If the melt temperature is too high per the documented setup sheet investigate why. Evaluate the barrel temperature set points, the actual back pressure, and the screw recovery speed. If any of these are not running to the documented process the result can be overheating of the material. Determine that all actual machine variables meet the specified set points.

44.3.2 Stringers Troubleshooting Mold Issues

Potential mold-related problems include:

- Hot tip temperature
- Tip-bearing surface
- Lack of cooling
- Valve gates
- Tip orifice size

44.3.2.1 Mold: Hot Tip Temperature

On hot runner molds without valve gates the tip temperature must control the freeze off of the plastic at the tip. If the tip temperature is set too high or the tip is overheating the result is likely stringing. Hot drop temperatures can be verified with a long thin probe that can be inserted into the drop to measure temperatures within the drop (see Figure 44.3). Always ensure proper personal protection equipment is worn and use appropriate safety precautions when checking temperatures.

Check that the hot runner temperatures are set correctly based on the documented process. Verify that the actual hot runner temperatures are reading correctly. Avoid running hot runners without thermocouples, because simple percent current output settings will result in inconsistent heating of the zone.

Positioning of hot runner thermocouples can lead to overheating of sections of a hot runner. A well-designed hot runner system will provide uniform heating across the entire manifold. Areas of heat sink can cause overheating due to inconsistency in the manifold. If there is too much contact area on a drop the mold steel will act as a heat sink and the zone may have to run hot to avoid freezing off.

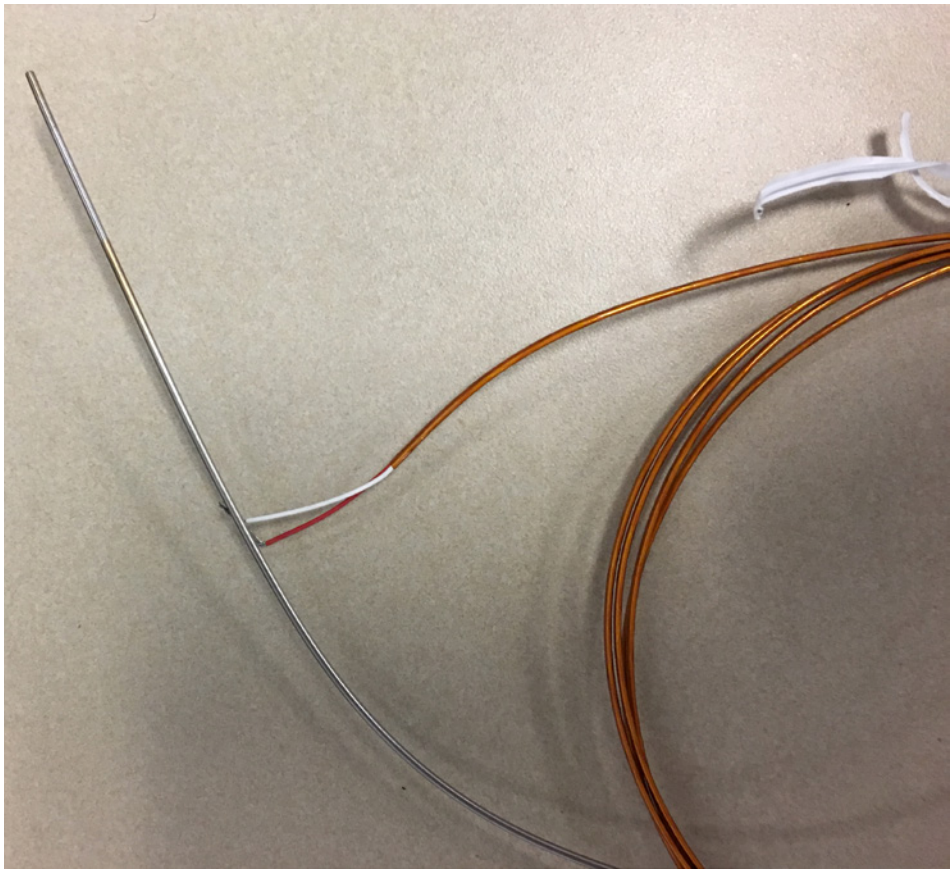


Figure 44.3 Thin thermocouple used for probing inside a hot drop

44.3.2.2 Mold: Tip-Bearing Surface

The amount of contact that the hot tip has with the mold is called bearing surface. This contact between the mold and tip will impact the temperature of the hot tip. If there is not enough bearing surface contact with the mold the tip will run hot and result in drool or stringers. Too much bearing surface contact can lead to excessive freeze off of the tip so bearing surface is very important to consider when troubleshooting hot tip related problems. Also keep in mind with valve gates that the only cooling that the valve pin will experience is when it is in the closed position contacting the bearing surface at the shut off with the tip.

44.3.2.3 Mold: Lack of Cooling

Molds should be designed with cooling around the tip locations. Without adequate cooling, the plastic in the tip will not reach a freeze point and will either drool or string. Evaluate cooling designs prior to mold build to ensure that the drop area is being cooled.

44.3.2.4 Mold: Valve Gates

If a hot runner system has valve gates it is critical that the valve pins reach a full forward position to ensure that the gate is closed. If the pin is not traveling all the way forward the heat at the tip will tend to create stringing problems.

Verify if the valve pins appear to be traveling forward. If a valve gate is having stringing defects it is time to take the mold to the tool room to determine why the pins are not reaching a full forward position. The cylinder may have worn seals or if the pin is adjustable it may be out of adjustment. Also determine if the pressure (pneumatic or hydraulic) is adequate for the pin to actuate completely.

44.3.2.5 Mold: Hot Tip Orifice

The larger a hot tip orifice is the more difficult it will be to cool. When there is a large mass of plastic at the hot tip the plastic acts as an insulator and can restrict how well the plastic cools, which in turn can lead to stringing problems. There have been cases where going with a smaller tip orifice led to a reduction in stringing problems.

44.3.3 Stringers Troubleshooting Machine Issues

A few basic issues on the machine can lead to stringers including:

- Nozzle heater setup
- Machine performance
- Nozzle tip

44.3.3.1 Machine: Nozzle Heater Setup

A key part of injection molding is maintaining accurate and repeatable temperatures of the melt delivery system. An often-underappreciated part of the machine is the nozzle and various adapters required for the nozzle (see Figure 44.4).

Evaluate how well the nozzle is covered with heater bands and thermocouples. If the nozzle is long and only has a thermocouple located in the wrench flats of the nozzle the balance of heat from end to end on the nozzle can be problematic. If using long nozzles and adapters make sure that they have separate thermocouple

control zones. By maintaining multiple zones of control on nozzles and adapters thermal control will be improved.

Figure 44.4 Extra-long nozzle assembly with multiple heater bands

Sometimes a nozzle tip must be isolated thermally from the mold to reduce stringers. The typical way to create this isolation is to add a piece of cardboard between the nozzle tip and the sprue bushing. There are also molded insulators that can be installed on the tip to help as well as Kevlar fabric insulators.

44.3.3.2 Machine: Machine Performance

See Chapter 8 on machine performance.

44.3.3.3 Machine: Nozzle Tip

Verify that the correct nozzle tip orifice and type are being used. The nozzle tip information should always be clearly documented during process development to ensure future repeatability. If the wrong nozzle tip is used the material may not freeze off the same as it previously had. If the nozzle tip and sprue are extra thick they may not cool well leading to stringing.

44.3.4 Stringers Troubleshooting Material Issues

The main causes of material-related stringing are:

- Moisture content
- Volatiles

44.3.4.1 Material: Moisture Content

If a material is molded with excess moisture the viscosity of the material will drop and the gas pressure from the volatiles will tend to push the material from the nozzle or tip. Gas generation in the material can lead to high residual pressures on the shot, which will tend to cause the material to drool or string.

Check the moisture content on the material. If the material is not adequately dried the process will need to be shut down to allow for proper drying.

To see more specifics on drying refer to Chapter 9.

44.3.4.2 Material: Volatiles

Many things in the material can result in volatiles in the melt stream such as:

- Low molecular weight components
- Lubricants
- Degraded regrind

Any of the above items might cause gas to evolve from the material. These volatiles are gaseous in the melt stream and just like with moisture can create a residual pressure on the shot in the barrel. The gas pressure can force the material to drool or string.

If the defects are isolated to a specific lot of material, engage the material supplier to determine possible solutions. If running regrind try running with 100% virgin material to see if the problem is improved. If the issue is linked to regrind check for fines in the regrind that may be melting quickly and degrading.

45 Voids

45.1 Description

Voids are internal pockets inside the part wall stock. Voids are different from bubbles even though they may appear the same. Bubbles are trapped gas in the part but a void is actually a vacuum that was created from shrinkage. To distinguish a bubble from a void slowly warm the wall stock with a heat gun. If the area sinks it was a void, but if it blisters up it was a gas bubble. Figure 45.1 shows a void in a polycarbonate part.

Also known as: internal sinks

Mistaken identities: gas trap, bubble, blister

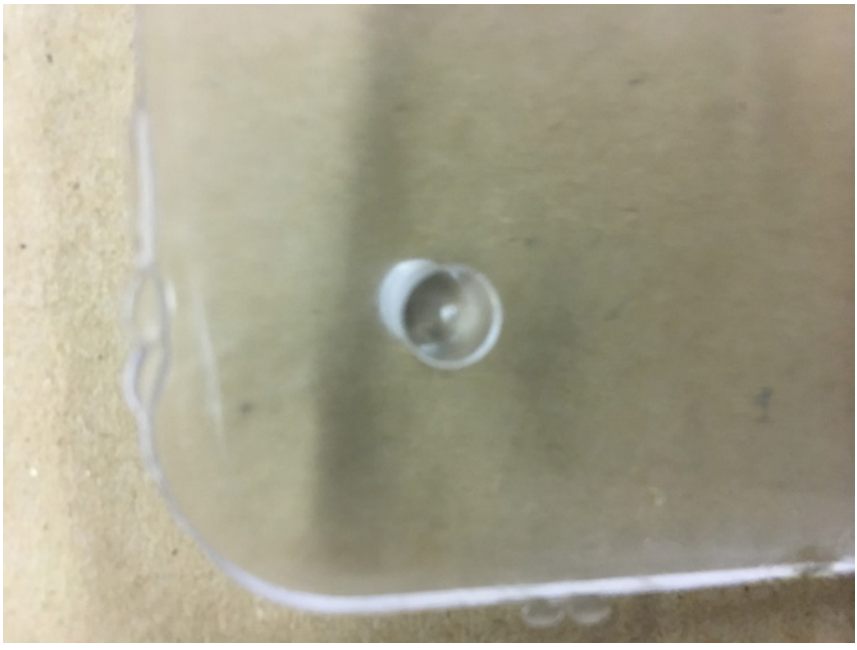


Figure 45.1 Void in thick section of part

45.2 Voids Troubleshooting Chart

Table 45.1 shows the voids troubleshooting chart.

Table 45.1 Voids Troubleshooting Chart

Molding Process	Mold	Machine	Material
low second-stage pressure	cooling	machine performance	
short second-stage time	thick sections		
low or high mold temperature	gate location		
high melt temperature			

45.3 Voids Troubleshooting

Voids are often overlooked because they are not visible on an opaque part. Transparent parts will visually show voids; they will most often appear in thicker wall stock areas. Voids may lead to a decrease in physical properties on the molded part so it may be necessary to cut a cross section of the parts to ensure the process is not producing voids.

Design can play a major impact on voids. If nominal wall stock is not maintained or if a part is gated thin to thick the likelihood of a void forming will rise. Many of the same concerns in part design that will be sink makers can also lead to voids. Trying to process around a poor part design will lead to narrowed process windows that will raise the cost of production on a part.

45.3.1 Voids Troubleshooting Molding Process Issues

Molding process issues impacting voids are:

- Low second-stage pressure
- Short second-stage time
- Low or high mold temperature
- High melt temperature

45.3.1.1 Molding Process: Low Second-Stage Pressure

When plastics are heated to a melted state the distance between the molecules expands due to the increased energy in the plastic. During the cooling phase of the process the distance between these molecules contracts leading to molded part shrink. When this shrink occurs in some part details the result can be formation of a void.

Second-stage pressure is the part of the molding process that compensates for the plastic shrink. Enough cavity pressure must be applied to limit the amount of shrink, which in turn can restrict formation of voids.

Confirm that the machine has been set up correctly per the documented process. If the process is running at the correct set points look for opportunities to increase second-stage pressure. If the mold has cavity pressure transducers verify that the cavity pressure is matching the established process template.

Increasing second-stage pressure will lead to a more packed out part that is less likely to have voids. If increasing pressure watch for dimensional changes in the parts; higher second-stage pressure will create a larger part. Increased second-stage pressure may also lead to sticking issues due to over-packing. When troubleshooting it is important to STOP and think through the potential negative impacts that the process adjustment may cause.

45.3.1.2 Molding Process: Short Second-Stage Time

If second-stage time is set too short the gate will not freeze, which can result in material flowing back out of the cavity. If cavity pressure is not maintained the plastic will shrink at a higher level which can result in voids in thick areas of the part. Voids near the gate location are a very big indicator that the gate is not sealed.

Verify that a gate seal study has been completed for the mold. Evaluate the set second-stage time against what is required to achieve gate seal. On parts that are to be processed with gate seal ensure that the second-stage time is set approximately 10% longer than the required time for gate seal because this will ensure that the gate is sealed even when conditions vary. As an example, if the gate seals at 10 seconds the second-stage time should be set at 11 seconds.

45.3.1.3 Molding Process: Low or High Mold Temperature

Low mold temperature can lead to increased pressure drop across the cavity. This increase in pressure drop can result in less effective cavity pressure being applied throughout the cavity. Low mold temperatures will remove heat faster from the molded part and can result in faster freezing of wall stock and gates. If the process does not provide adequate time to effectively pack out thick sections the part will either have sinks or voids.

High mold temperatures can lead to increased levels of shrink in the molded part. The increase in shrink can result in voids forming internally in the part. Mold temperature can be a tradeoff between increased pressure drop due to a cool mold and increased shrink rate in a hot mold. Balancing the heat input against the heat extraction during the molding process is a key step to producing quality parts.

Part ejection temperature and mold surface temperature will provide key information on whether the mold temperature is running correctly. These are details that should be documented during process development to provide troubleshooting information. If the mold surface or part ejection temperature is running at the wrong temperature verify the following:

- Thermolator set point and actual water temperature
- Flow rate of coolant
- Correct water circuit layout

  Proper cooling is critical to providing a repeatable part. Ensure that the process conditions match the established process.

45.3.1.4 Molding Process: High Melt Temperature

A higher melt temperature will lead to a longer time to cool the part. This increased cooling time can create a void when the outer wall of the part freezes well against the mold but the interior plastic continues to cool and shrink. Depending on how resilient the frozen wall is will impact whether the part forms a sink or a void. If the frozen wall is stiff and will not move due to the internal stresses created by differential cooling the plastic may form a void.

Check the process melt temperature against the documented process. If the melt temperature is high investigate the potential causes of the increased melt temperature. Many factors can impact melt temperature including:

- Barrel temperatures
- Screw recovery rate
- Back pressure
- Screw design

All of the above factors should be compared to the intended process settings to validate that the process is running correctly. If anything is not running correctly make the required adjustments to bring the process back to the intended settings.

45.3.2 Voids Troubleshooting Mold Issues

Mold-related causes of voids may include:

- Cooling

- Thick sections
- Gate location

45.3.2.1 Mold: Cooling

Locations in the mold that are not cooled effectively will run hotter than the rest of the mold which can lead to increased shrink of the molded part. When this shrink occurs in a thick wall section the part may either sink or it may have a void.

Verify the part ejection temperature matches the documented process. Examine the mold after running the process to determine any locations of hot spots. Verify that the mold is watered correctly and has adequate flow in all circuits.

For further information see Chapter 14 on mold cooling.

45.3.2.2 Mold: Thick Sections

Although it is more of a part design problem, thick sections will be discussed as part of the mold design. When parts have thick cross sections, there will be a tendency to form either sinks or voids. This tendency is due to increasing shrink rates in thick sections. Thick areas cool longer and may not be packed out as well because surrounding thin wall stock may freeze before the thick area is packed out.

During part design every effort should be made to minimize thick areas on the part. Coring of thick areas will help to minimize the opportunity for voids to form.

Thick cross sections have a negative impact on cycle time simply because it takes longer to cool the thick sections of the part. Plastic is an excellent insulator that will keep heat from conducting through thick sections resulting in longer required cooling times. To maximize the cooling efficiency of a mold, thick sections must be cored out to try to maintain nominal wall stock.

In some cases thick wall sections will require the gate to also be increased to ensure that the thick section is fully packed out. Increased gate size will lead to a longer gate seal time which in turn leads to a longer cycle time, which means lower profit potential.

45.3.2.3 Mold: Gate Location

If a part is gated into a thin wall and has thick sections away from the gate the odds of voids will be increased. Nominal wall stock should be maintained whenever possible. If a gate is located in a thinner wall the thin wall will freeze before the thick section is packed out, which will result in increased shrink in the thick section.

Whenever a part is designed with nonuniform wall stock the gate should be located in the thicker section to ensure that the thick area is packed out. Always try to core out the thick sections first; however, if they cannot be cored try to locate the gate so that it will have maximum impact on the areas of concern.

Computer flow simulation can provide a frozen layer plot that will highlight areas that are freezing prematurely. Try to use the software to locate the gate in a location where the thick sections of the mold can be packed out.

45.3.3 Voids Troubleshooting Machine Issues

See Chapter 8 on machine performance.

45.3.4 Voids Troubleshooting Material Issues

There is a big difference between molding of a transparent or an opaque material concerning the detection of voids. When molding opaque materials, it is not possible to see a void, but in a transparent material a void will be obvious. To find a void in an opaque material the part must be cross sectioned to detect the void.

In critical molded parts X-ray inspection may be required to detect voids. This method provides a nondestructive means of examining the molded part.

46 Warp

46.1 Description

Warp is a defect where the part is distorted from the intended shape. Figure 46.1 shows a straight edge laid against a warped part, notice the gap between the part and straight edge.

Also known as: distortion

Mistaken identities: none

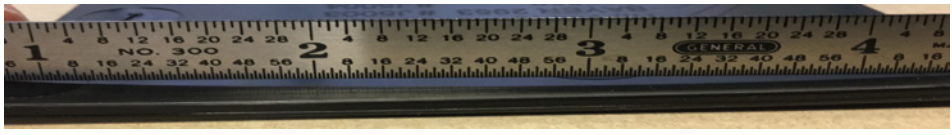


Figure 46.1 Example of warp

46.2 Warp Troubleshooting Chart

Table 46.1 shows the warp troubleshooting chart.

Table 46.1 Warp Troubleshooting Chart

Molding Process	Mold	Machine	Material
mold temperature	sticking	machine performance	material type
melt temperature	cooling issues	robot handling	part packaging
pressure differential	cavity imbalance		filler content and type
gate seal	gating		
low cooling time			
post-mold handling			

46.3 Warp Troubleshooting

Warp can be a challenging defect to resolve. By the very nature of heating and cooling a thermoplastic there will be shrinkage and stresses that can lead to warp. The most common reason for warp is differential shrink throughout a molded part. This differential shrink can come from a number of factors including:

- Differential cavity pressure
- Differential cooling
- Differential orientation especially with semi-crystalline materials

Each of the above factors is covered specifically below in each appropriate section.

When it comes to warp, part design drives many of the problems encountered in molding. The key design rules for plastic are absolutes when it comes to avoiding warp. The four design rules that are most important are:

1. **Maintain nominal wall stock**

   Plastics will shrink differently depending on the nominal wall stock. Wall stock impacts how far a part will fill and pack from a gate, cooling rate, and orientation. If a part has varying wall stock the plastic will have different shrinkage rates, which will result in warp. Trying to process around a fundamental design flaw will result in manufacturing process windows that are unacceptable for producing quality parts.

2. **Avoid gating thin to thick**

   If nominal wall stock varies (see #1) do not locate a gate in the thinner wall stock regions. Locating a gate in such a region will result differential shrink because the ability to pack out the thick section will be driven by the thin wall. Once the thin wall freezes the thick areas can no longer be packed, which results in an under-pack situation in the thick regions resulting in increased shrink. Another result of gating thin to thick is the thin areas may be overpacked by trying to pack out sink in the thick areas, which again leads to differential shrink and warp.

3. **Rib to wall ratio**

   To avoid thick hot areas around ribs design standards for a rib to wall ratio should be followed. Typical standards would be a 40–70% thickness for a rib depending on the material. This will help minimize sink but also helps avoid a hot thick junction at the base of the rib that can lead to warp due to excess shrink. Thin ribs will freeze faster than the nominal wall and may lead to warp opposite of the rib location.

4. **Utilize rounded corners**

   Try to avoid sharp corners whenever possible. A rounded corner with an appropriate radius will allow even filling and cooling. Also sharper corners will create

stress concentrations, which in turn can lead to warp as the plastic tries to relieve stress during cooling.

Many warp problems can be significantly improved during part and mold design. Addressing the above concerns during part design is a foundation to provide warp-free parts. Modern molding CAE software packages can help to predict warp and should be used as an iterative tool to evaluate design impact to warp.

Another factor to consider is that even if a part is stiff enough to resist warping after ejection from the mold the part may end up with high internal stress. Internal molded-in stress may lead to failure of the part especially when subjected to environmental stress cracking conditions because the environmental stress crack resistance (ESCR) of a part is impacted by the molded-in stress.

46.3.1 Warp Troubleshooting Molding Process Issues

Molding process concerns that can lead to warp include:

- Mold temperature
- Melt temperature
- Pressure differential
- Gate seal
- Cooling time
- Post-mold handling

46.3.1.1 Molding Process: Mold Temperature

An important factor relative to warp of a part is the heat deflection temperature (HDT) of the material being molded. When a thermoplastic is above its HDT the material will not maintain its shape when exposed to loading. If a molded part is ejected at a temperature above the HDT the part will be easily distorted.

Plastic cooling rate is one of the key process variables in injection molding. The mold ends up acting as a heat exchanger to remove the heat energy from the molded part. To bring the molded part temperature down to levels where the part can be ejected without excess warp requires the mold temperature to be correct. Every material will have a recommended mold temperature which is a vital reference point for any process. Questions should definitely be asked if the mold is running at 180 °F but the supplier recommends 80–120 °F for the mold temperature.

The most obvious contributor to mold temperature is the set point on the temperature control unit or thermolator as they are commonly called. The water temperature will help determine what the mold temperature will run at with the actual steel temperature normally running warmer than the water set point. Actual steel

temperatures should be measured on the mold and documented for future reference and troubleshooting. When checking mold temperatures make sure that the mold has been running on cycle for long enough for the steel to reach its true running temperature. Studies have shown that this can easily take an hour or longer of run time, depending on the size of the mold and the cooling layout. See Figure 46.2 for an example of how a mold heats up during the cycle. Both temperature curves are recorded with mold thermocouples. The flat horizontal part of the curves on the far left is the baseline mold temperature and the steel temperature increases as the molten plastic touches it.

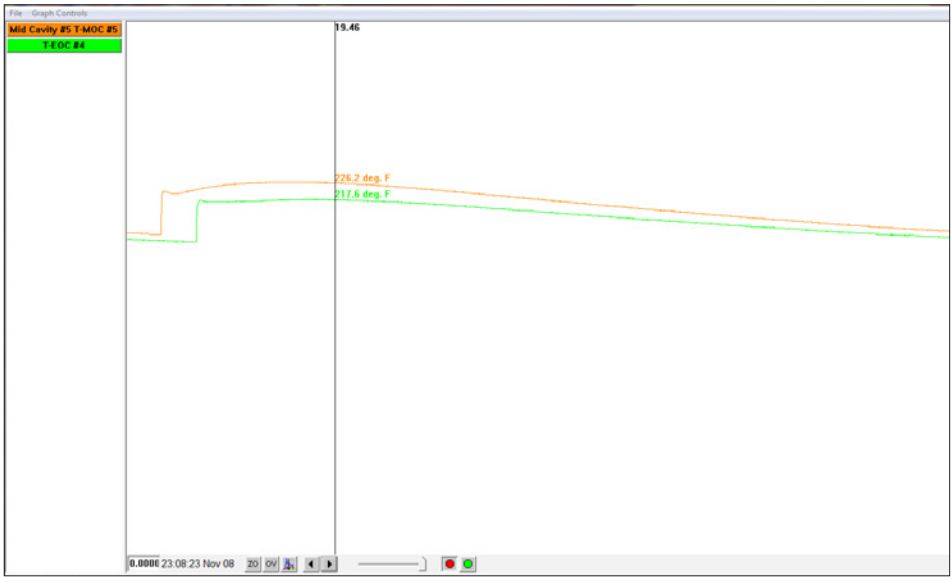


Figure 46.2 Process monitoring curves for mold thermocouples

One of the key impacts of getting the melted plastic to a temperature below its HDT is the cooling efficiency of the mold. To ensure maximum cooling, turbulent water flow through the mold needs to be achieved. To achieve appropriate water flow, the thermolator must have a sufficient amount of pumping capacity to adequately reach the desired flow rate through the cooling channels. Water flow is another parameter that must be documented during process development, so that during troubleshooting the original flow can be compared to the current flow.

Another important check on mold cooling is the inlet water temperature versus the output water temperature. Depending on how critical the molded parts are, the maximum temperature rise for the water should be less than 4 °F or 2 °F on critical jobs [1]. This temperature can be checked with a handheld pyrometer with a surface probe: place the probe right on the water fitting to check temperature. If

the outlet water is increasing above the recommended maximum the cooling circuits should be evaluated for the opportunity to split single long flow lengths into multiple circuits.

The temperature of the part after ejection should also be documented during process development. The optimum way to do this is with a thermal imaging camera which provides an accurate overall view of the temperature of a part (see Figure 46.3). Another way to check this part out temperature is by using a pyrometer with a surface probe. If using the surface probe method, document where the temperatures were taken to ensure repeatable results.

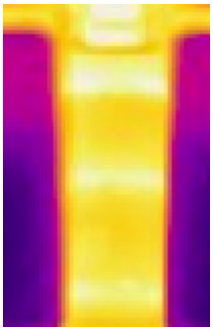


Figure 46.3 IR thermal image of part

When troubleshooting cooling-related causes of warp check the following items:

- Water temperature
- Water flow rate
- Water temperature input versus output
- Verify same capacity thermolator (pump capacity, horsepower, and cooling valve size)
- Confirm mold has had water lines correctly routed

Note that plastic will warp toward heat. The classic example is often referred to as the trash-can effect. Picture a square or rectangular polypropylene trash can; the sides will very often be warped inwards. The trash-can effect indicates where the plastic warped toward the hotter core half of the mold. This type of situation will be very typical of box-shaped cross sections because the inside corners of the core will normally be harder to cool and will run hotter. The warmer mold surface will cause increased shrink on that side of the part, which in turn leads to increased shrink and the tendency to pull the part that direction. When designing box shapes into molded parts extra effort should be applied to designing maximum cooling into the core half of the mold.

The fact that plastic warps toward heat can be used as an advantage in some parts. By running an intentional mold temperature from cover to ejector it is possible to drive warp in the direction opposite of what it normally would go. Use caution when setting intentional water temperature differential because the steel will expand differently depending on the temperature. Depending on the shut off conditions in the mold too much offset can lead to galling and mold damage. Check with a tooling engineer or mold maker to determine how much temperature differential is permissible for the mold in question.

Case Study: Cooling Warp

In this case a talc-filled polypropylene was being molded in a part that had a rectangular section that had a tendency to warp inwards. If the part warped inwards it created assembly problems with the mating components. A water circuit was run through the mold surrounding the rectangular section. This outer water line was run at 120 °F and the rest of the mold was run at 80 °F. When running with this water offset the rectangular section would actually want to warp outwards rather than inwards, which resulted in a part that had no assembly issues.

46.3.1.2 Molding Process: Melt Temperature

High melt temperatures can lead to a hotter part at ejection. In injection molding the management of the heat energy put into the plastic and its removal is critical. If the plastic is hotter and all other process conditions remain the same, the part will be hotter when ejected, which in turn may result in increased warp.

Changes in melt temperature will also impact the pressure across the cavity. If the melt temperature is cooler than prior molding runs the pressure drop will increase across the cavity. This increased pressure drop can lead to higher cavity pressure differentials that result in differential shrink resulting in warp. The worst-case scenario would be a melt temperature that is so low that there is a short shot (0 psi cavity pressure) at end of fill.

Verify that the melt temperature matches the established process. Uncontrolled high melt temperatures could lead to increased cooling time to compensate. Document melt temperature and verify when troubleshooting.

46.3.1.3 Molding Process: Pressure Differential

Another process factor that impacts warp is differential cavity pressure. When plastic is heated the distance between the molecules increases but during cooling the plastic contracts, which results in shrinkage. The amount of shrink of plastic will change based on how much pressure is used to “pack” the molecules together.

When a part has areas that have differences in cavity pressure the shrink will vary in these areas. Variation in shrink across a part will result in warp because the areas of higher shrink pull the part resulting in warpage.

A challenge in molding is minimizing the pressure drop across the cavity and trying to generate as uniform of a cavity pressure as possible. By minimizing differences in cavity pressure, the impact of differential pressure will be minimized. One key to minimizing pressure drop is filling the mold fast because this helps maintain a more consistent viscosity and provide a more uniform cavity pressure.

Distance from the gate has a major impact on cavity pressure: the farther away from the gate the lower the pressure. Evaluation of the gate location is critical before a mold is built; if the end of the cavity is too far from the gate it will be impossible to provide a uniform cavity pressure. In many cases adding additional gates to provide better pressure distribution can be the key to success. This is another problem that CAE flow analysis can provide a great deal of help with.

Wall thickness will also play an impact on cavity pressure distribution especially when combined with distance from gates. Thinner wall areas will freeze faster and may restrict the ability to provide adequate and equal cavity pressures. This is especially important if the part has variation in wall stock. Wall thickness differences are further magnified when molding a semi-crystalline material because the impact on crystallinity will show as increased risk of warp.

Another factor to keep in mind is that areas that are not pressurized well against the mold cavity will tend to shrink away from the mold quicker than areas with higher cavity pressure. Once the plastic has shrunk away from the walls of the mold the effective cooling rate will decrease due to the lack of contact with the mold (remember the mold is serving as a heat exchanger). Improved pressure distribution and higher overall cavity pressure can lead to more efficient cooling, thus reducing warp.

Verify that the second-stage pressure is set correctly. Confirm that the transfer from velocity control to pressure control is responding well. The machine should transfer quickly into the set second-stage pressure and do so consistently. If the machine does not respond well the material may start to freeze prior to effective pressurization of the cavity.

46.3.1.4 Molding Process: Gate Seal

There are times when running with gate seal will lead to warp problems. When running with gate seal the plastic that is packed into the cavity will be held in the cavity. Gate seal can result in locking in excess pressure at the gate area of the part. This is more common on center direct-gated parts where high local pressure radiates from the gate.

In some cases running without gate seal will allow the plastic near the gate to equalize with the rest of cavity. This equalization will help reduce the pressure differential in the cavity. Another concern is on a rectangular lid-style design that has different flow and pack distances as the plastic travels away from the center gate. A center gate on this shape of part will guarantee that there are areas filling at different rates as well as packing differently.

If a part is warping during process development try running with gate seal and without gate seal to determine if there is any impact on the warp. If the warp is improved by running without gate seal and all other factors are acceptable (sink, voids, gloss, and dimensions) run the mold without gate seal and document as such. The assumption that every mold has to run with gate seal is wrong.

When troubleshooting a production problem with warp verify whether the mold is supposed to run with or without gate seal. Check the process to determine that the second-stage time is set correctly. If in doubt that the gate is sealed or not conduct a gate seal study (see Chapter 3 on the Decoupled Molding™ Process).

46.3.1.5 Molding Process: Cooling Time

When a part is removed from the mold too soon warp can develop. A molded part must be allowed to cool long enough to allow the plastic to reach a temperature below the heat deflection temperature (HDT) of the material. If a part is ejected too soon the plastic will still be pliable and can easily deform resulting in a warped part.

Verify that cooling time and overall cycle time are consistent with the established and documented process. If the time is wrong make the appropriate adjustment to bring the process back to the baseline. Check the part ejection temperature to determine if the part is not being cooled as effectively because a higher part temperature after ejection can indicate cooling problems that need to be addressed. Adding cooling time to compensate for a mold cooling issue is a costly means of working around the problem: always try to drive to the root cause rather than process around a problem.

46.3.1.6 Molding Process: Post-Mold Handling

Sometimes warp occurs during and/or after part removal from the mold or during handling by the robot or operator. Utilize the STOP process to observe potential causes that are impacting the part during or after ejection.

Look for situations where the operator is stacking parts on top of each other prior to packing. Also make sure that the parts are being packed correctly because improper pack can lead to warp. Another thing to keep an eye on is how quickly the parts are being packed because in some cases the operator may be clearing the conveyor belt and packing parts that are hotter than normal.

Closely watch the robot to determine if the part is being damaged from the robot end of arm. There are cases where the robot actually pushes excessively hard against the part while picking and causes deflection of the part. Watch the robot to see if it is moving straight back away from the mold because sometimes a robot program will have a step that causes an indirect move that results in the part hanging up on the mold and becoming distorted. See Section 46.3.3.2 for more information on robot handling.

Case Study: Post-Mold Handling

In this case a small polypropylene automotive trim piece was being molded. The operator would occasionally find a couple of warped pieces as they inspected parts. Using the STOP method and Observing the parts after molding highlighted that parts were catching on the conveyor and becoming warped from the force of the conveyor pushing against them. The location on the conveyor where the parts were getting trapped was shielded and the warped parts were completely eliminated.

46.3.2 Warp Troubleshooting Mold Issues

Mold-related warp causes include:

- Sticking
- Cooling issues
- Cavity imbalance
- Gating

46.3.2.1 Mold: Sticking

If a part is sticking in the mold the part may distort when ejected. Parts that stick will require additional force to remove from the mold, which in turn may lead to distortion that appears to be warp.

Verify that the part is not sticking to the cover or the ejector half of the mold. Often cracking noises will accompany sticking parts. If the noise is heard during mold open, the part is likely sticking to the cover, but if the noise occurs during ejection then the part is sticking to the ejector half.

A great way to verify that a part is not sticking is to run a cycle in semi-auto and stop the cycle prior to ejection. Examine how the part sits on the steel of the ejector half of the mold. A part that has lifted away from the ejector steel can indicate that the part stuck in the cover half of the mold. Also try to manually remove a part from the ejectors and lifters because many times it is possible to feel where the

part is sticking when ejected. If sticking is detected the mold should be repaired rather than trying to process around the problem.

For more details refer to Chapters 35 and 36 on sticking.

Case Study: Warping Parts from Sticking

In this case a PC/ABS was being molded in a part that had to be consistently flat. If the part had any warp it would cause an assembly and functional concern at the customer. Many experiments were conducted focusing on process conditions and packaging, all without reliably resolving the problem. During this time the concerns at the customer continued to amplify.

During a mold sampling run the mold was stopped before the part was ejected. Examination of the part relative to the ejector half of the mold showed that the corners of the part were lifted away from the steel. A part was molded with mold release on the cover side of the mold and the mold was again stopped prior to ejection. With the mold release the part was tight to the ejector half steel. Both sample parts were measured for warp and the part molded with the release agent was perfect while the other part was out of specification.

Closer examination of the cavity showed that there was a slightly rolled parting line that was causing the part to stick slightly in the cavity. This rolled edge was cleaned up with a polishing stone and samples were molded. The resulting samples showed no warp. Checking the parting lines for burrs and rolled edges was added to the preventative maintenance routine for this mold, and the warp problem never reoccurred.

Following the STOP method may have led to faster resolution of this problem. 4M troubleshooting would have helped to ensure that the mold was evaluated for problems rather than trying to rely on the process to solve the problem.

46.3.2.2 Mold: Cooling

Uniform temperature is critical to avoid warp on molded parts. The mold acts as a heat exchanger and removes the heat energy from the molten plastic. The mold should be designed with optimized cooling; unfortunately, cooling is often a tradeoff for ejection and other tooling actions.

See Chapter 14 on mold cooling for more details.

46.3.2.3 Mold: Cavity Imbalance

On a multicavity mold it is critical that all cavities experience the same process conditions for the plastic. If a multicavity mold has imbalance between the cavities the process conditions within the cavities will not be the same. The resulting differences can impact many process variables including:

- Cavity fill rate
- Cavity pressure

- Cavity cooling rate
- Gate seal

If the above conditions are not kept consistent between cavities warp is an expected potential outcome. It is absolutely critical for a multicavity mold to be as balanced as possible. If the mold is not balanced and some cavities warp and others do not the mold will have to be modified to ensure balance.

For more details on cavity balance see Chapter 12.

46.3.2.4 Mold: Gating

Warp can be impacted by the gates on a molded part. Gates need to be evaluated in regards to the overall part to try to provide adequate fill and uniform pressure distribution within the cavity.

If a part does not have enough gates the flow length becomes excessive for the material and can lead to a stalled flow front and lack of fill. Just as important a consideration is the distance from the gate that the pressure is applied during pack. Pressure will vary across the cavity as the distance from the gate increases and this variation in pressure can lead to warp. Adding a second gate to a part may allow for a much better distribution of pressure and reduce the warpage of a part.

Another factor relating to flow length and packing length is where the gate is located on a part. During mold design it is important to consider the flow length and how the pressure will be distributed through the cavity. This is another area that doing a CAE evaluation of the part can yield very valuable insight when looking at "Sink Index" and "Frozen Layer" plots.

It is important to keep in mind that if a gate is too thick the part may become overpacked near the gate. This uneven packing can cause warp on the part, especially if the rest of the part is freezing while the gate area is still being packed. There have been many cases, especially with semi-crystalline materials, where going to a thinner gate reduced warp on a part.

46.3.3 Warp Troubleshooting Machine Issues

Machine factors influencing warp include:

- Machine performance
- Robot handling

46.3.3.1 Machine: Machine Performance

As with many other defects, knowing that the machine is actually achieving its set points is important to avoiding warp.

See Chapter 8 on machine performance.

46.3.3.2 Machine: Robot Handling

When a robot removes a part from a mold there is a possibility of warping the part. If the robot pushes the part against the mold with enough force it can cause distortion of the part resulting in a warped part.

Another concern with robot handling is how the robot extracts the part from any mold details or components such as lifters. If the robot does not clear the mold component as it retracts, the part can actually catch on the mold component and become warped as the robot causes the part to deflect away from the mold component.

If the robot handling of the part is suspected as the root cause of an issue, try running a cycle in semi-automatic mode and hand removing the part from the mold. If the part that was removed by hand does not have the defect it may be a good indicator that the robot extraction is the problem.

46.3.4 Warp Troubleshooting Material Issues

Material factors can have a significant impact on warp including:

- Material type
- Part packaging
- Filler content and type

46.3.4.1 Material: Material Type

The number one difference for warp impact is between amorphous and semi-crystalline materials. The orderly crystalline structure of semi-crystalline materials will lead to more contraction of the space between the molecules during cooling. When semi-crystalline materials cool in the mold the resulting shrink will be higher due to the way the crystallinity builds in the material. Increasing amounts of crystallinity will lead to increased amounts of shrink and greater likelihood of differential shrink. In the melted state semi-crystalline materials will actually be in an amorphous stage, and freezing the material in more of an unordered amorphous state will lead to reduced shrinkage. Keep in mind if using a cold mold to minimize crystallinity the part may experience warp at a later point in time when it is exposed to temperature higher than the mold temperature.

Semi-crystalline materials will also tend to experience anisotropic shrink, which means that the shrinkage in the flow direction will be different to the shrink across the flow direction. This can often lead to differential shrink, which results in warp that is difficult to predict. Injection molding CAE software can help predict this anisotropic shrink and allow a molder to cut a mold with differential shrink rates. When using a semi-crystalline material make sure to account for the increased shrink and warp during part and mold design.

Materials with fillers will shrink less. The difference between a filled polypropylene and an unfilled could be 0.015 in/in shrink or more. The higher the shrink potential of the material the more likely it will be that there are regions of differential shrink resulting in warp. Using a filled material where appropriate can help to reduce shrinkage and as a result improve part warp (see Section 46.3.4.3).

When running a fiber-filled material, orientation will potentially lead to anisotropic shrink. Fibers will establish an orientation relative to flow direction with the fibers lining up in the direction of flow. This orientation can result in a restriction of shrink in the direction parallel to flow whereas the shrink perpendicular to flow occurs at more normal rates.

46.3.4.2 Material: Part Packaging

This is really a handling issue but the packaging that is used for a part may contribute to warp of a part. If parts are packed in a manner that applies force to the warm molded part the result can be part warp. This may result from packaging that does not fit the part well, or bulk packing parts that are too warm; also layer packing can put weight on lower layers resulting in warp.

Selecting the appropriate pack is as important to providing a quality part as any other part of the process. Pack selection during process development must be evaluated to keep a part protected through storage and delivery to the customer. Make sure that the part fits the packaging without being flexed or bent. Evaluate if bulk packaging is appropriate for a given part because many times the part must be better protected than what bulk packaging provides.

46.3.4.3 Material: Filler Content and Type

Fillers will reduce the amount of shrink that a part experiences. If a material has a large amount of filler variation or does not have the correct loading of filler the resulting increase in shrink could lead to warp.

Verify that the correct material is being used along with the correct filler content. Many times the material supplier will provide filler content on the certificate of analysis for the filled material.

If using an unfilled material review if a filled material would be acceptable and evaluate the impact on warp. It is a simple experiment to run some filled material in a mold and compare the results to the unfilled material. If the filled material is better investigate the cost, appearance, and impacts on dimensional and physical properties. After verifying all important factors work with the customer to switch materials to the filled version.

It is also important to note that fillers will act as nucleating agents and affect the way semi-crystalline materials crystallize. This effect can be positive or negative but must be factored in when troubleshooting warp. Nucleated materials typically will not experience as much shrink as would a non-nucleated material.

Keep in mind that when using a fiber-filled material the orientation of the fibers will result in anisotropic shrink that can in turn lead to warp. The shrink of fiber-filled plastic in the direction of flow will normally be less than the shrinkage across flow. Fibers will stay oriented after the mold has been filled and this orientation can in turn lead to warping parts. This orientation impact must be considered during part and mold design. One factor that can help this anisotropic shrink is to combine mineral fillers and fiber fillers: the mineral filler will help restrict the cross flow shrink thereby providing less differential in shrink.

References

[1] Bozzelli, John, “Dont Neglect Cooling...There’s Money To Be Made”, *Plastics Technology*, April 2010

47 Weld Lines

47.1 Description

When plastic flows around an obstruction in a mold the point where the material comes back together is called a weld line. If the material continues to flow together after coming together it will become a meld line. Plastic will always form a weld line when it splits in a mold. Figure 47.1 shows an example of weld lines.

Also known as: knit lines

Mistaken identities: scratches, flow lines, cracks

Figure 47.1 Weld lines

47.2 Weld Lines Troubleshooting Chart

Table 47.1 shows the weld lines troubleshooting chart.

Table 47.1 Weld Lines Troubleshooting Chart

Molding Process	Mold	Machine	Material
fill velocity	venting	clamp tonnage	filler type and content
melt temperature	wall stock	velocity to pressure transition	viscosity
second-stage pressure	gate location	machine performance	
mold temperature			

47.3 Weld Lines Troubleshooting

Weld lines can cause visual defects but can also impact the physical properties of the molded part. Optimizing weld lines is important to the overall quality of the product. During the design phase it is critical that everyone understands locations where weld lines will occur. Designers must also understand that the processor cannot change the laws of physics and that splits in the melt stream will cause weld lines. Use of mold-filling software will help to establish the location of weld lines (see Figure 47.2). A weld line plot is a useful visualization to show the designer and to help them understand where the weld lines will occur and the potential severity of them.

The weakest weld lines will be where the material comes together and stops flowing. If flow occurs after the material comes together the resulting (now called) meld line will tend to look better and be stronger.

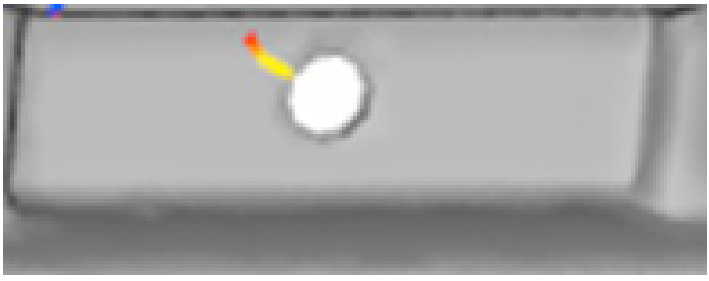


Figure 47.2 Moldflow plot of weld line

47.3.1 Weld Lines Troubleshooting Molding Process Issues

Some of the molding process conditions that impact weld lines are:

- Fill velocity
- Melt temperature
- Second-stage pressure
- Mold temperature

47.3.1.1 Molding Process: Fill Velocity

In general, if the mold is well vented faster fill velocities will lead to better weld lines. A common issue is when the mold is not well vented and technicians will slow down fill velocity, which will impact the weld line quality. If weld lines can be "slammed" together they will typically be stronger and less visible.

Also with fast fill times the material will maintain a more consistent viscosity and allow for improved ability to pack the weld line with second-stage pressure. Maintaining consistent viscosity during first-stage fill helps minimize the pressure drop across the cavity, which allows a more consistent cavity pressure during second stage.

Verify that the fill velocity is optimized. Look for venting improvements that may lead to improved fill rates. Lack of venting is not a good reason to reduce fill velocity; instead make the tooling adjustments to allow for optimized processing.

47.3.1.2 Molding Process: Melt Temperature

The plastic melt temperature needs to be high enough to allow the weld to have some bonding across the weld. If melt temperature is too low the weld line will be weak because the molecules will not have a strong attraction.

Fountain flow is the description of how plastic flows from the inside out when filling through a mold wall section. The advantage of fountain flow is that hot material is always coming to the front of the flow, which helps provide hot enough material to a weld line. Results from flow analysis show that in some cases melt temperature rises during fill and in other cases it decreases during fill. It is critical to provide an adequate melt temperature to allow the weld line to "bond" together well. Figure 47.3 is a magnified photo of a weld line that shows a valley where the flow fronts welded together.

Figure 47.3 Magnified weld line

Verify melt temperature against the documented process as well as against the supplier-recommended settings. If melt temperature is off, verify that the various settings that contribute to melt temperature are set correctly. The key items to check relative to melt temperature include:

- Screw recovery speed
- Back pressure
- Barrel temperatures
- Residence time
- Screw design (only changes if the screw was replaced or the job was moved to another machine)

If any of the above conditions have changed when compared to the documented process make the required correction.

47.3.1.3 Molding Process: Second-Stage Pressure

It is important to provide adequate pack pressure at weld lines. Higher pack pressures will force the material at the weld line to co-mingle and provide stronger bonds between the plastic at the weld line. Fully packed-out parts will provide stronger weld lines.

Verify pack pressure to the documented process. Make sure to account for intensification ratio as required. If weak weld lines are a problem try increasing the second-stage pressure to check for improvement.

47.3.1.4 Molding Process: Mold Temperature

Normally higher mold temperatures will allow for better formation of a weld line. With a warmer mold the plastic will pack the knit easier and the appearance of the weld line will reduce. Using a higher mold temperature will allow better chances for the molecules to become entangled at the weld line.

Increasing the mold temperature will improve the appearance and strength of many weld lines. If weld lines are a concern try sampling the mold at a higher temperature.

There are rapid mold heating systems on the market that can provide a much higher mold temperature. These technologies will do a nice job of eliminating the visual aspect of the weld line and strengthen the weld line. If weld lines are of critical concern investigating a rapid mold heating technology may be the best solution.

47.3.2 Weld Lines Troubleshooting Mold Issues

Key impact items for the mold include:

- Venting
- Wall stock
- Gate location

47.3.2.1 Mold: Venting

Venting is one of the main culprits of bad weld lines. When two flow fronts meet, the gas that is traveling ahead of the front must be vented to provide a high-quality weld line. If gas is trapped between the flow fronts the weld line will be weak and visually obvious. The gas acts as a buffer between the flow fronts and inhibits the chance for the material to actually bond together.

Even with meld lines poor venting can lead to visual defects. There are cases when what appears to be a flow line will appear along a part. Running short shots will sometimes show that these lines are actually a meld line that is occurring from a portion of the melt front hesitating. The initiation of this defect can sometimes be seen as a slightly skewed flow front. Running short shots is a very important troubleshooting tool that will help identify the evolution of many defects. These flow front hesitations can occur as a portion of the melt stream runs along a radius while the rest of the flow front travels in the nominal wall.

Venting problems are one of the most common mold-related problems that people will process around. There have been many cases where “can’t add more venting” was proven wrong and the defect was resolved after adding more venting. Venting correctly does not cause a mold to flash! Remember that vents have a depth measure but also a width measure. Figure 47.4 shows a weld line about to form: this area must be vented to produce an adequate weld line.

Figure 47.4 Short shot around a boss: weld line will form as the part fills out

When troubleshooting a weld line problem, a good place to start is with an evaluation of the venting. Make sure the mold is clean and all vents are open. If the mold has alternate venting methods such as sintered metal inserts make sure that they are still working, and replacing an insert may solve the problem.

See Chapter 7 on venting for more details.

47.3.2.2 Mold: Wall Stock

It is a given that any time the melt front splits around part details such as a core pin or insert a weld line will form, but what can be overlooked is the influence of wall stock on the flow front. Thin areas of the mold may cause hesitation for a portion of the melt front resulting in a weld or meld line when the flow fronts rejoin.

Maintaining uniform nominal wall stock is rule one when designing for plastics and this is one of the reasons why.

If a weld or meld line is occurring in an area that does not make sense check short shots to look for flow front hesitation. Computer flow analysis will help identify these issues on part designs. Sometimes the wall stock may have been unintentionally thickened in an area which in turn leads to a meld or weld forming.

47.3.2.3 Mold: Gate Location

The location of the gate as well as the number of gates can impact weld lines. If a part is gated with multiple gates the part will have weld lines between each of the flow fronts (unless sequentially valve gating). It is important to keep the weld lines in mind when designing gates, and a flow analysis software can help with predicting all potential weld line locations.

Sometimes during mold design, it is possible to position a gate so that any weld lines forming from mold cores will be in a noncritical location. Look for opportunities during design to try to influence the impact of weld line location. It is critical to understand the customer expectations for weld line appearance, location, and strength.

Sequential valve gates can be used with multiple gates on a part to provide a part without visible weld lines from gate flow fronts. Sequential valve gating is when an initial start of fill valve is open and downstream gates are not opened until the flow front reaches them. This avoids the weld line that is formed from the flow fronts from multiple start of fill gates.

47.3.3 Weld Lines Troubleshooting Machine Issues

Potential machine impacts on weld lines can come from:

- Clamp tonnage
- Velocity to pressure (V to P) transition
- Machine performance

47.3.3.1 Machine: Clamp Tonnage

As mentioned in Section 47.3.2.1 venting of a mold is critical to formation of quality weld lines. If the mold is clamped with excess tonnage there is a good chance that the vents will hob shut. Too much tonnage on a mold literally moves the steel resulting in a lack of venting.

Inspect the clamp tonnage setting to determine if the clamp force is set correctly. Just because a machine has 500 tons of clamp force does not mean that it is appro-

priate for every mold that runs in the machine. Sometimes smaller molds will run in a machine due to machine capacity, shot size, ejector stroke, or mold open stroke. When running a small mold in a machine evaluate if the tonnage should be reduced. If the process is established with less than maximum clamp tonnage make sure that the machine is running with the correct clamp force.

To check for over-tonnage on a machine reduce the clamp tonnage setting or back off the die height setting on a toggle. If the weld lines seem to improve this is a good indicator that more venting will help the situation. If the reduced clamp tonnage provides an acceptable part either the process can continue to run with reduced tonnage or ways to improve the venting on the mold have to be sought.

47.3.3.2 Machine: Velocity to Pressure Transition

Velocity to pressure transfer is what occurs when the machine changes control from first-stage velocity control to second-stage pressure control. If this transfer does not occur in a crisp and quick manner the delay can cause a stall in the flow fronts. When flowing plastic stalls, the resulting weld lines will be weaker than normal. This factor is often overlooked but can be critical for ensuring weld lines are of the highest quality level.

Velocity to pressure transition is discussed in detail in the Chapter 8 on machine performance.

47.3.3.3 Machine: Machine Performance

If the machine is not achieving the desired set points weld lines will become a more challenging problem to correct.

See Chapter 8 on machine performance.

47.3.4 Weld Lines Troubleshooting Material Issues

Some potential material concerns are:

- Filler type and content
- Viscosity

47.3.4.1 Material: Filler Type and Content

Fillers in the plastic material will not travel across the weld line interface. Because of this materials with fillers will usually have more visible weld lines. The plastic at the weld line will usually not have as much of the filler present, which in turn will lead to a variation in appearance between the surrounding plastic and the weld line (see Figure 47.5). Notice in Figure 47.5 how the fiber orientation changes at the weld line as well as the decreased level of fibers on the surface.

Some fillers will establish orientation as the plastic travels through the mold. This orientation may change direction at the weld line leading to a more obvious weld.

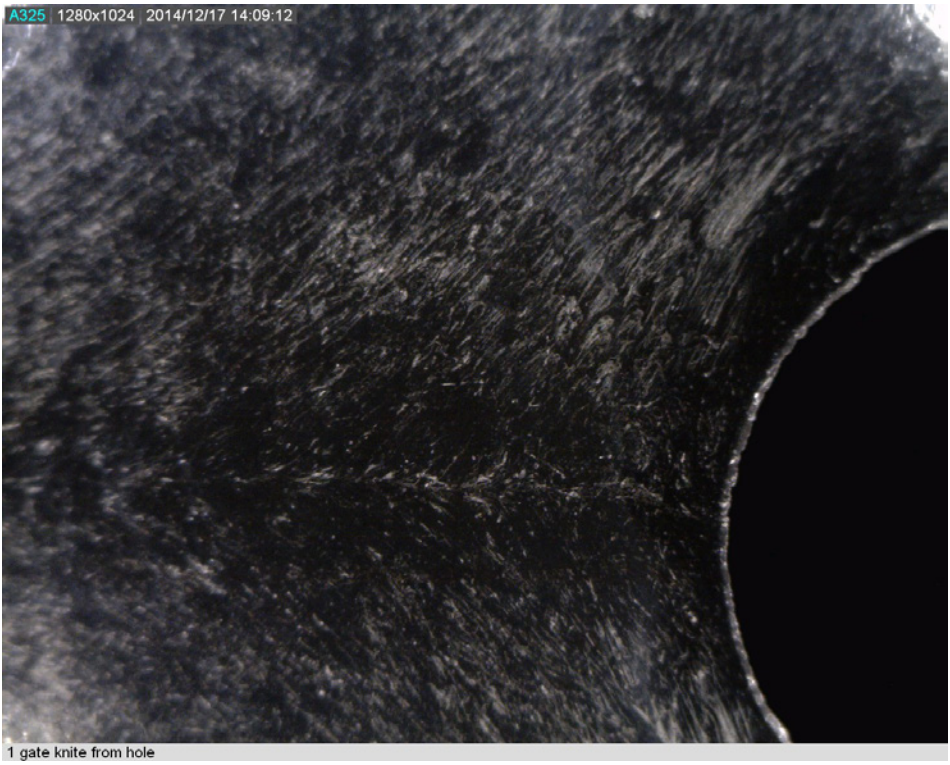


Figure 47.5 Weld line on a glass-filled material

Unfilled materials will give the best weld lines; adding fillers will result in more obvious weld lines. Sometimes the solution to objectionable weld lines is to turn them into a meld line, which can be achieved by adding an overflow to let the material flow together after welding together.

Case Study: Metallic-Filled Materials

In this case an ABS was being molded with a metallic flaked colorant. The metallic flakes had a tendency to orient with flow which created variation in appearance depending on the flow direction. Another major issue was that whenever a weld line formed the metallic flakes would go from aligned with flow to aligned perpendicular to flow. The flakes at the weld line with a change in orientation led to a very obvious weld line and the very resin-rich plastic right at the weld line also contributed to a visual difference. See Figure 47.6 for a magnified view of a weld line on a silver metallic material. ▶

Ultimately this problem was never completely resolved. Different-sized flakes with an aspect ratio of closer to 1 : 1 (height to width) were used, which helped to improve the defect. Metallic flakes will create struggles with weld lines. If there is interest in a metallic-filled material make sure that everyone sees molded samples with weld lines before deciding on a color.

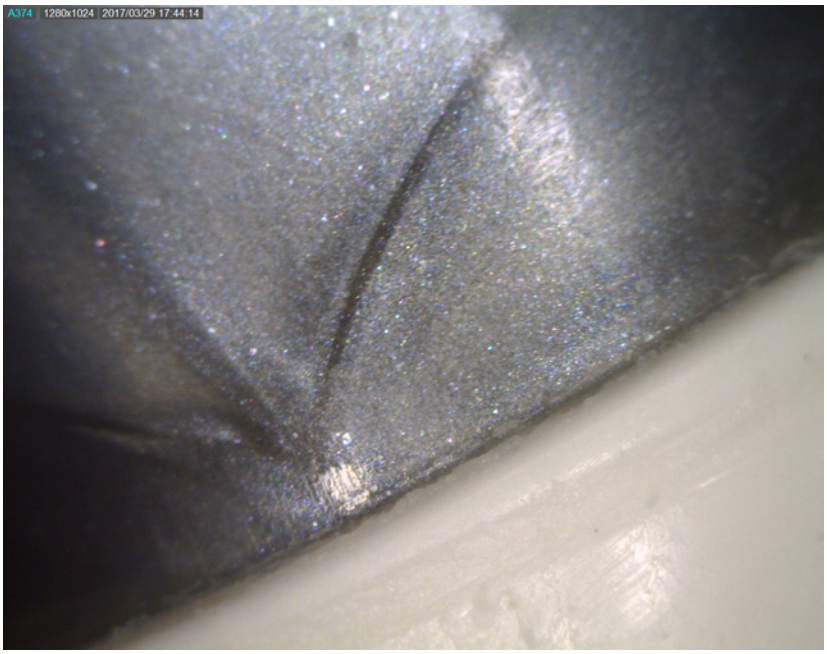


Figure 47.6 Weld line with metallic pigment

47.3.4.2 Material: Viscosity

A lower-viscosity material will provide a better chance for a weld line to come together and allow entanglement of the plastic molecules at the weld line. Lower material viscosity will allow easier mold filling and better transfer of pressure across the cavity. A higher melt index material may result in better weld lines. Remember that increased melt index often means reduced physical properties. Be sure to validate any material change through the customer and all applicable testing.

With hygroscopic materials watch for viscosity shifts from moisture content. A change in moisture content can lead to a viscosity difference that impacts the formation of weld lines.

Index

Symbols

4M 4
4M Basic 8 7, 301, 383
4M method 18, 32
4M troubleshooting 148, 466
5 Why 5, 13

A

abrasive 441
ABS 37, 81, 144, 149, 179, 217, 219, 237, 328, 415, 436, 479
acetal 155, 171, 189, 221, 235, 247, 272, 395
acid flush 130, 404
acrylic 328
actuated vent pins 59
additive feeder bins 211
additives 81, 190, 296, 328, 416
agglomerates 220
airflow 85
air poppets 341, 425
alarms 383
alloys 280
aluminum oxide 48, 49
amorphous 251, 252, 266, 344, 468
analog input 123
anisotropic 468, 470
anisotropic shrink 253
anti-seize 160, 216
ASA 328
aspect ratio 480

B

backfill 58, 381
backfilled 301, 302
back pressure 76, 156, 167, 175, 183, 201, 203, 229, 266, 311, 324, 383, 403, 415, 417, 419, 454, 474
baffle 404, 405
ball-style non-return valve 250
barrel 135
barrel heaters 137
barrel shot capacity 258
barrel temperatures 153, 167, 183, 188, 266, 311, 403, 454, 474
bead blasted 442
bearing surface 55, 97, 447
Beaumont Effect 104
Beaumont Inc. 104
Beaumont, John 104
bimetallic carbide barrels 391
black/brown streaking 91, 209
black or brown specks 131
black specks 91, 186, 209
blades 259
blast media 47
blend 303, 326
blisters 165, 236, 451
blowing agent 412, 427, 429
blush 141
bore 42
bosses 340, 347
Bozzelli, John 29, 61, 85
brainstorming 15
brass tools 340
breakaway distance 262
breakaway speeds 339
breaking 223
brittle 225
brown streaking 137, 151, 181, 186
bubblers 128, 259, 366, 405
bubbles 81, 165, 296, 451
buildup 130, 173, 186, 194, 324, 341, 350, 387

burning 120, 311, 402
burns 53, 101, 181, 258
burrs 228, 340, 350
byproducts 186

C

CAD model 303
CAE 269
CAE flow analysis 463
CAE software 459
calibration 75, 273
capacity issues 255
caprolactam 179
carbon 220
carbon buildup 131, 135
Carl Fischer titration 86, 416
cashew gate 34, 108, 148, 235, 277, 279, 359, 408
cavity 242, 340
cavity balance 30, 31, 101, 116, 360, 385, 395, 407
cavity cooling 103
cavity fill balance 107
cavity fill rates 103
cavity fill time 115
cavity imbalance 33, 466
cavity instrumentation 111
cavity locking angles 214
cavity pressure 59, 103, 243, 258, 267, 282, 290, 292, 338, 453, 462, 473
cavity pressure curve 361, 408
cavity pressure transducers 111, 241, 321, 347, 399
cell packed 373
cellulosic 81
change log 17
changeover 189, 193, 196, 210
changeovers 200
charred 183, 190
check valves 64
chicken scratches 299
chiller 403
chute 226, 372
clamp force 106, 282, 285
clamp open 263
clamp parallelism 293
clamp pressure 282
clamp tonnage 285
clamp velocities 262
cleaning 324
cleanout 231
cloudiness 91, 191
cold slugs 277, 436
cold slug well 147, 269, 423
cold starts 213, 389
colorant 179, 307, 442
colorant concentrate 161
color changes 96, 194
color concentrate 163, 190, 201, 204, 231, 237, 238
color mixing 203, 271, 419
color streaking 91, 199
color swirls 181, 199, 209
component deflection 247
compression ratio 161, 171, 187, 426
computer-aided flow analysis 98
computer flow analysis 304, 477
computer flow simulation 456
condensation 420
containers 196
contaminants 138, 162, 185
contamination 91, 157, 172, 191, 196, 202, 206, 222, 231, 233, 237, 391, 395, 427
conveyor 226, 263, 371, 372
cooling 103, 106, 125, 246, 256, 257, 326, 351, 366, 454, 455, 460, 466
cooling capacity 403
cooling circuits 404
cooling rate 115, 326, 458, 459
cooling time 242, 244, 256, 258, 260, 261, 348, 358, 434, 464
cooling tower 403
copper alloys 366, 438
copper-based alloys 128, 405
core 226, 259, 294
core circuit 294
cored out 367
core half 226
core pin 261, 406, 476
core pressure 290
core pull cylinders 290
core pulls 44
cores 128, 340
coring 455
cost 255
cover half 225
cover side detail 338
crack 169, 170, 194, 424
cracking 223
crazing 227
crystallinity 246

cushion 64, 157, 245, 249, 383, 401
cushion monitoring 383, 388
cycle time 125, 201, 226, 244, 246, 255, 295, 419, 421, 439
cylinder bore gauge 392
cylinders 41
cylinder size 290

D

damage 215, 340, 350
decompression 168, 184, 245, 415, 418, 445
Decoupled II® 32
Decoupled III® processing 122
Decoupled® II molding process 258
Decoupled®/scientific molding 29, 30, 243
deep draw 261
deep rib 340
deflection 282, 364
deformation 358
degating 277
degradation 151, 167, 225, 338, 417, 426
degrade 183, 190, 212, 229, 441
degraded 91, 131, 201, 387
delamination 91, 209, 231, 233, 414
depression 397
desiccant 82–84, 416
desiccant beds 84
design 145, 227, 297, 303, 307, 313, 349, 351, 367, 398, 406, 452, 455, 458, 472, 477
Design of Experiments 16
dew point 82, 83, 416, 442
dew point meter 83
dew point monitors 23
D-gate 35
dial indicator 285
diamond 194
diamond chrome 177
diamond polish 51
die blue 366
die height 285
die locks 214
dieseling 181
differential cavity pressures 364
differential cooling 247
differential shrink 261, 364, 366, 458
dimensional issues 72, 101, 247
dimensions 239, 242, 283
dings 369
direct gating 95
discoloration 151
dispersion tip 272
distort 338
distortion 347, 357, 457
donuts 374
downtime 186, 343
draft 44, 261, 340, 359, 373
draft analysis 340
draft angles 228, 340, 349, 352
drag 359, 373
drags 228
draw polish 51, 341, 349, 359, 374, 435, 436
drool 168, 184, 418, 445, 450
drooling 97
drop 212
dryer monitoring 123
drying 81, 415
drying hopper 210
dull 141, 309, 319
durometer 342
duty cycle 273
dynamic check valve study 64

E

eDART® 400, 409
EDM 228, 349, 387, 408
ejected 258
ejection 225, 226, 261, 279, 464
ejection force 359
ejection speed 348
ejection stroke 261, 350, 372
ejection surface area 349, 359
ejection temperature 455, 464
ejector area 374
ejector box 289
ejector pins 112, 230, 279, 365
ejector plate 263, 289, 351, 360, 365
ejector plate stop buttons 365
ejector side 342, 356
ejector speed 357
end cap 135, 187, 215, 236
end of arm tooling 217, 370
end of fill 183, 302, 310
end of fill cavity pressure 271
end-of-fill transducer 114
environment 253
environmental stress cracking 227, 459
erosion 185, 288, 290, 350, 440, 441

F

family molds 109, 292
feed lines 211, 237, 264
feed throat 170, 215, 264, 404, 415, 420, 426
fiber orientation 478
filled material 251, 252
fillers 81, 307, 362, 469, 478
fill only 63, 143, 310
fill only shot 29, 243, 250, 284
fill only weight 30, 72, 267, 301, 322, 379, 400, 410
fill pressure 246
fill time 258, 267, 306, 310, 381, 402
fill velocity 142, 183, 242, 258, 265, 266, 300, 310, 322, 333, 473
filters 85
fines 139
first stage 243
first-stage fill 284
first-stage injection velocity 381
first-stage velocity 402
fishbone diagram 4, 14
flaking 277, 424
flame-retardant 328
flame retardants 179
flash 54, 72, 101, 258, 279, 281, 297
flow 259
flow analysis 303
flow analysis software 246, 321
flow defects 307
flow distance 246
flow front 302, 303, 333, 475, 478
flow front velocity 312
flow leader 303
flow length 103, 253, 268, 292, 302, 313, 380, 467
flow lines 141, 299, 414
flow marks 141
flow meter 27, 123, 402
flow pattern 303
flow rates 327, 454
foreign material 220, 249, 269, 386
fountain flow 34, 473
fracture 223
freeze 245
frozen layer 384, 467
frozen layer plot 246, 456
full taper tip 270

G

galling 214
gallons per minute 402
gas assist 293, 425
gas assist control 122
gas assist injection molding 411
gas booster 411
gas counter pressure 429
gas traps 165, 181, 184, 451
gate 141, 152, 260, 267, 303, 313, 477
gate design 108
gate flaking 420
gate freeze 268
gate land 34, 268
gate land length 146, 334
gate location 268, 334
gates 253, 279, 386, 395, 406, 467
gate seal 30, 33, 38, 114, 146, 147, 227, 241, 256, 322, 399, 400, 463
gate seal study 29, 322, 453
gate seal time 433, 455
gate size 142, 245, 334, 407
gate spacing 246
gate style 380
gate thickness 145
gate vestige 34
gating 33, 257, 313
gating thin to thick 458
gaylord 218
gaylord tipper 394
gel tape analysis 220
general-purpose screw 202, 203
gibs 214
glass 252
glass bead 48
glass content 316
glass fibers 309, 310
glass-filled 33, 34, 39, 50, 59, 128, 249, 259, 307, 309, 311, 328, 391, 436, 440–442
glass-filled materials 168
glass transition temperature 348
glassy 309, 310
glassy surface 315
gloss 47, 53, 101, 142, 319, 363
gloss variation 72, 309, 328
glossy 319, 355
grained 320
grease 157, 185, 214
grinder 139
grinding 163

gripper fingers 352
grippers 352, 371
Groleau, Rod 29, 61, 111
guided ejection 374
gusset ribs 439

H

halo 141
handling 227, 369
hang up 96, 135, 152, 189, 194, 195, 204, 210, 215, 226, 235, 236
hang up areas 160, 212, 215
heat deflection temperature 459
heater 95, 99
heater band 159, 184, 448
heater band burnout 159
heat exchanger 404
heat sink 323
heel block 290
hesitation 299, 302, 476
hesitation lines 300
high fill pressure 265
high-gate 35
high-gloss 328
high melt temperature 132, 225
high pressure 268
high temperature 174
hob 290
hold volume 399, 409, 410
hopper 222, 237, 394
hopper loader 210
horn pins 44, 290
hot drops 34, 40, 387, 446
hot drop tips 96, 423
hot oil 312
hot runner 93, 95, 146, 169, 189, 194, 212, 226, 235, 246, 261, 268, 303, 383, 422, 446
hot runner cables 388
hot runner controller 120, 247
hot runner leakage 98
hot runner manifold 120, 289, 401, 422
hot runner plugs 154, 388
hot runner system 153, 202
hot runner temperatures 134, 313, 388, 405, 420
hot runner tip 146
hot sprue 438
hot tip 91, 386, 443, 448
hydraulic 186
hydraulic cylinders 236, 350
hydraulic gauge 294
hydraulic leaks 293
hydraulic pressure 294
hydraulic pressure gauge 76
hydraulic response 74
hydraulics 41
hydraulic system 306
hydrolysis 90, 230, 274, 296, 315, 441
hygroscopic 89, 171, 189, 197, 416, 441, 480

I

imbalance 101
impact modifiers 81, 232
impinge 334
infrared (IR) 77
injection compression 60
injection fill velocity 175
inserts 128, 405
insulator 445, 448, 449, 455
insulator caps 96
insulator gaps 97, 158
intensification ratio 167, 203, 338, 382, 475
IR temperature gun 201
IR thermal imaging 137
is/is not 17
ISO pipes 128

J

jetting 34, 141, 192, 331

K

Kistler Group 112
knit lines 184, 312, 471
knock-out bars 263, 351, 360

L

L*a*b* 207
label 206, 222, 237
labeling 196
laminar flow 104, 126
land length 145
larger tonnage machine 283
laser welding 288
L/D 187
L/D ratio 426

leader pin 372
leakage 250
leaking 236, 245, 383
leaking hot runner manifolds 389
letdown ratio 204, 412, 307
lifter 185, 229, 247, 277, 279, 348, 350, 374
lifter pocket 366
lifter read 363
lifter rods 365
linear thermal expansion 253
line of draw 290, 340
loaders 211, 420
loading of pack 373
load sensitivity 63
lock angle 41
locking angle 290
lockout/tagout 21
lockup 288, 295
long flow lengths 257
long glass fibers 311
long glass-filled 394
loop 299
loss on weight 86, 416
low back pressure 168
low-gloss 328
low-vestige-style hot tip 268
low vestige tip 96, 386
lubricant 214, 236
lubricants 179, 264, 353, 361, 421, 424

M

machine controllers 75
machine hopper 211
machine shutdown 132
machine testing 61
machining/cutter marks 358
machining marks 349
magnets 220, 272
maintenance 70
manifold 212
marbling 199
marring 374
masterbatch 179, 412, 442
material change 186, 193, 212
material contamination 131
material feed 243, 394, 415
material leaking 401
mechanical switches 45
meld lines 300, 303, 312, 471
melt consistency 269
MeltFlipper® 104
melt flow index 232, 296
melt front 266
melt index 480
melt probe 201, 311
melt stream 167
melt temperature 123, 144, 167, 174, 183, 192, 201, 225, 234, 243, 256, 264, 266, 286, 301, 311, 323, 338, 348, 357, 384, 403, 415, 445, 454, 462, 473
metal 269
metal inserts 220
metallic flaked 479
metallic flakes 307
metal shavings 404
microcellular foam 429
microetching 48
micro texture 323, 325
milkiness 191
mineral 252, 328
mineral deposits 404
minimum mold size 295
mismatch 135, 160, 169, 187, 194, 195, 212, 215, 235, 236, 291
miter locks 134
mixing 201
mixing elements 161
mixing nozzles 202, 270, 391
mixing screw 161, 203
mixing style screws 202
mixing tips 148, 202
moisture 81, 86, 171, 178, 189, 197, 415, 450
moisture analyzer 23, 86, 343, 416
moisture content 230, 274, 306, 315, 343, 441
mold breakaway speeds 227
mold cleaner 157, 215
mold cleaning 185, 214
mold component deflection 364, 365
mold cooling 244
mold deflection 25, 285, 289
mold design 169, 253, 257, 327, 334, 349, 350, 365, 366
Moldflow® 321, 333
mold inserts 157
mold open speed 339
mold release 344
mold steel temperature 128
mold surface 320
mold surface finish 324
mold surface temperatures 302

mold temperature 119, 120, 144, 193, 244, 246, 256, 311, 323, 347, 357, 365, 384, 402, 453, 459, 475
mold temperature control unit 122
molecular weight 90, 232, 274, 296, 307, 343
MuCell® 293
multicavity 101, 292, 341, 385
multimeter 22
multiple ejector strokes 263

N

nippers 371
nitrogen 411
nominal wall 452, 455, 458, 475
nominal wall stock 257, 261, 340, 367, 406, 477
non-return valve 64, 245, 249, 379, 383, 389, 401, 410, 426
nonuniform 201
nonuniform wall stock 257
nozzle 135, 148, 187, 215, 236, 390, 439, 448
nozzle adapter 215
nozzle drool 435
nozzle leak 392
nozzle lengths 271
nozzle orifice 236
nozzle temperature 144, 444
nozzle tip 169, 215, 248, 269, 383, 390, 401, 410, 418, 420, 423, 439, 443, 449
nozzle tip orifice 135, 391
nucleated 197, 412
nucleating agents 252, 264, 469
nucleation 362, 442
nylon 39, 86, 87, 89, 97, 171, 179, 180, 231, 272, 275, 288, 306, 315, 394, 415, 441
nylon 6 312, 317
nylon tip 270

O

oil 185, 214
opaque 197, 456
operator 372
orientation 458, 469, 470, 479
orifice 97, 98
orifice gauge 148
outgassing 53, 190
overflow 59, 312, 411, 479
overflows 122
overheated 201
overheating 78, 96, 187
over-packed 347
over-packing 338

P

pack 283
packaging 372, 469
packed 242
pack rate 115
part design 224, 261, 340, 359, 367, 374, 398
part design issues 257
part detail 359
part ejection temperature 128, 365, 366
parting line 280, 282, 286, 287, 340, 374, 383, 444, 466
parting line bearing surface 425
parting line flash 185
part out temperature 258, 357, 385
path of least resistance 381
Paulson, Don 29
PC/ABS 34, 81, 144, 149, 161, 203, 221, 235, 274, 415, 466
peak injection pressure 379
peak pressure 274
perimeter venting 425
perpendicular 381
piano black 324
pigments 207
pin deflection 349
pin marks 355
pin push 72, 261, 355, 359
pin read 363
pins 277
plastication 175, 177, 264, 274
plasticizers 190
plastic pressure 203
platen 289, 293
platen deflection 106, 289
platen parallelism 106
platen wrap 295
plate out 173, 385
plugged 269, 404
PMMA 171
poka-yoke 237
polish 194, 320, 321, 324
polished 50, 228
polishing 47, 288
polycarbonate 81, 133, 149, 171, 179, 328, 343, 352, 415, 423, 441, 451

polyester 315, 343, 352
polyesters 81, 171, 230, 415, 441
polyethylene 155, 237
polypropylene 38, 90, 132, 146, 155, 193, 197, 207, 237, 272, 341, 359, 366, 373, 425, 436, 462, 465, 429
polystyrene 196
poppet valve 402
porous steel 58
post-gate cavity pressure 400
post-gate transducer 114
post-mold handling 464
post-molding 224
post-mold shrinkage 243, 244
PP 35, 204, 205
PPO 179
preload 290
preloaded push pins 342
pressure 76
pressure differential 243
pressure drop 103, 115, 146, 202, 246, 266, 270, 321, 379, 403, 453, 462
pressure-drop study 38, 39
pressure gauge 273
pressure limited 29, 38, 39, 86, 203, 265, 266, 306, 310, 379
pressure limiting 66
pressure loss 34, 96
pressure profile 399
pressure relief valve 43, 294
pressure response 286
pressure-sensitive film 440
preventative maintenance 427
Priamus System Technologies 112
process control 122, 426
process monitoring 23, 71, 117
process monitoring curves 379
process monitoring system 76
process window 225
profile 235
profiled first-stage fill velocity 333
projected area 283, 292
proximity switches 45
pullers 358
pull sink 408
purge 91, 132, 342
purging 163, 168, 186, 200
purging compound 92, 196
PVC 132, 155, 171, 189, 190, 221, 235, 441
pyrometer 22, 99, 106, 120, 365, 460

R

radius 224, 228, 406
rapid mold heating 475
rapid mold heating and cooling 323
read-through 261, 320, 326, 363, 365, 366
record grooves 299, 300
recording 299
recovery speeds 156
regeneration heaters 84
regrind 95, 138, 163, 177, 196, 219, 231, 264, 271, 297, 307, 394, 450
release 339
release agents 179, 190, 232, 264, 344, 353, 361, 442
relief 54
residence time 132, 155, 187, 189, 207, 225, 235, 415, 419, 474
resin remover 342
restriction 271
return air 84
reverse-flow thermolator 169, 424
Reynolds number 126
ribs 347, 381
rib to wall 398
rib to wall ratio 458
ring gate 37
ring vent 421
rippled surface 323
RJG Decoupled III® process 394
RJG Decoupled Molding® 284, 379, 400
RJG Decoupled® process 382
RJG eDART® 23, 74, 84, 113, 203, 243, 273, 286, 410
RJG Technologies 111, 122
robot 78, 217, 226, 263, 352, 370, 465, 468
root cause 1
rubber gels 220
rubber modifiers 328
ruby stone 374
runner 242, 246, 260, 268, 297, 313
runner length 380

S

sand-blasted 320, 324, 325, 342
sand/glass blast 324
scale buildup 123, 404, 434
scientific molding 30
scrapes 279, 369
scrap recording sheets 14

scratch 300, 414
scratch and mar 320
scratches 369
screw 135, 161, 171, 187, 201, 426
screw design 425, 454, 474
screw L/D 161
screw recovery 201, 417
screw recovery rate 324, 454
screw recovery speed 167, 175, 403, 474
screw rotate 258
screw rotate delay 201
screw rotate speed 266
screw rotational speed 183
screw speed 229
scuffing 340
scuffs 370
second-stage pressure 225, 230, 240, 283, 284, 321, 347, 356, 364, 382, 399, 433, 453, 474
second-stage time 241, 322, 399, 453
second-stage velocity 305
second-stage volume 284, 305
self-locking cylinder 43
semi-crystalline 97, 197, 244, 251, 252, 264, 269, 344, 347, 395, 403, 412, 426, 442, 463, 468
sequentially valve gated 303
sequential valve gates 95, 477
sharp corners 224, 228
shear 146, 156, 175, 184, 229, 235, 417, 419, 427
shear imbalance 104
shear rates 38, 103, 270, 385
shear viscosity study 300
short shot analysis 300
short shots 53, 72, 101, 119, 166, 192, 272, 377, 383
shot size 245, 250, 258, 379, 419
shrink 226, 239, 246, 347, 399, 403, 406, 453, 455
shrinkage 283, 458, 463
shrink factor 252, 253, 283
shrinking 338, 339
shrink on 261
shrink rates 261, 412
shutdown 187
shut off 282, 287, 289
shut-off 44
shutoff 214
shutoff surfaces 134
silo 206, 222, 237
silver streaks 413
sink 72, 101, 261, 283, 402, 403, 412, 452, 454
sink index 467
sintered metal 476
sizing agents 316
sleeves 359
slide 185, 350, 374
slides 44, 229, 277, 279, 290
sliding ring 250
snapping 223
solenoid 406
solvents 227
splay 53, 90, 91, 98, 120, 137, 186, 197, 209, 231, 413, 421, 423, 445
spotted 366
spotting blue 22
spotty 319
spreader tip 97, 194, 269
sprue 147, 242, 246, 260, 268, 431
sprue break 434
sprue bushing 169, 215, 236, 383, 401, 434
sprue bushing seat 158
sprue opening 270
sprue picker 372
sprue puller 434, 436
sprue sticking 423
stalling 299, 303
startup 384
startup parts 302
steel temperature 327, 339
steel tools 288
stick 225
sticking 47, 72, 101, 185, 337, 342, 345, 347, 356, 358, 453, 465
sticking on cover 337
sticking on ejector 346
STOP 4, 18, 225, 252, 261, 280, 282
STOP analysis 288
STOP method 147, 370, 378, 466
STOP methodology 217, 303, 383
STOP process 146, 339, 348, 373
STOP troubleshooting 227, 231, 243, 246, 364
straight lifters 349
straight through sprue style 96
stratification 233
streakiness 191
streaking 199, 231
streaks 209
stress 227, 458, 459

stress concentrations 228
stress risers 224
string 450
stringers 288, 443
stringing 168, 184, 262
stuck plastic 387
styrene-butadiene copolymer 196
styrenic 204
sub gate 35, 36, 108, 235, 277, 279, 408
sub gated 359
suicide/crash condition 44
support pillars 289
surface area 42
surface finish 229
switches 45
systematic 3

T

talc 328, 462
TCU 128
Teflon thread tape 404
temperature 76
temperature control 425
template 382, 453
texture 47, 321, 324, 328, 342, 358, 373
texture depth 340, 374
thermal control 229
thermal image 259, 327, 403
thermal imaging 434, 461
thermal imaging camera 25, 106, 258
thermally stable 190
thermocouple 83, 95, 99, 147, 167, 184, 388, 422
thermocouples 76, 78, 111, 118, 156, 159, 229, 446, 448
thermolator 106, 257, 311, 365, 454, 459, 461
thermoplastic elastomers 50
thermoplastic polyurethane 50
thick wall sections 398
thin sections 302
thin wall 293, 303
thin wall sections 257
throughput 237
tie bar 293, 294
timing out 379
TiO_2 221
tip radius 440
toggle links 295
toggle machine 285
toggle wear 295
tonnage 292, 477
tonnage per square inch 292
TPE 341, 441
TPO 144, 149, 221, 328
TPU 90, 341–343, 352, 415, 417, 441, 442
tracer pellets 155
training 342
transfer 243, 322, 463
transfer position 250, 284, 322, 379
transfer pressure 301, 411
transferring 267, 305
transparent 171, 191, 456
trapped plastic 288
travel 229, 350
trimming 373
T-slot coupler 42, 44
tuning 70, 74
turbulent flow 126, 259, 351, 403
turbulent water flow 460

U

unbalanced flow 303
uncovered containers 237
undercut 228, 280, 339, 342, 435
uniformity 168
uniform melt 201
uniform nominal wall stock 240, 303
unmelted 250
unmelted pellets 269
unmelts 172, 272, 395
unmelt test 395
unsupported length 365
UV stabilizers 179

V

vacuum 166, 177, 263, 339, 341, 451
vacuum cups 217, 371
vacuum venting 58
valve gate 95, 96, 119, 247, 405, 447, 448
valve gate control 122
valve gating 477
valve pin 194, 405
valve pin actuation 303
valve tuning 410
vapor locks 402
velocity 65
velocity linearity 66, 314
velocity to pressure response 409

velocity to pressure transfer 72, 249, 286, 305, 382, 408, 478
velocity to pressure transition 361
vent 53
vent depth 54, 288
venting 106, 168, 175, 182, 184, 258, 288, 301, 311, 312, 327, 381, 385, 420, 421, 473, 475
venting ejector pins 56
vent inserts 58
vent land 176
vents 173
Venturi effect 169, 421, 425
viscosity 30, 183, 266, 274, 296, 307, 316, 393, 411, 450, 480
voids 101, 165, 397, 433, 451
volatiles 172, 173, 189, 450
volume 40, 72
volumetric additive feeders 204
volumetric feeder 211
volumetric flow rate 242, 272, 313, 379

W

wall stock 143, 197, 246, 253, 269, 302, 326, 476
wall stock transition 326
wall thickness 246, 292, 463
Ward, Norm 61
warp 101, 244, 247, 257, 258, 457
water circuit 454
water diagrams 106, 327
water flow 123, 127, 402, 404, 434
water flow rate 461
water leaks 100, 194
water line 169
water line layout 127
water temperature 127, 434, 461
water treatment 404
wear 213, 249, 258, 401
wear plates 134, 214
weld line 53, 59, 471
weld line plot 472
wet 427
white imprints 355
white streaks 191
witness line 305
wood-filled 328
worm tracks 331
worn barrel 391

X

X-ray inspection 456

Z

Zapox 194, 342
zero draft 340
zinc stearate 344, 362